HYSYS® 2004.2

Operations Guide

aspentech
driving process profitability

Copyright

October 2005

Copyright © 1981-2005 by Aspen Technology, Inc. All rights reserved.

Aspen Accounting.21™, Aspen ACM Model Export, Aspen ACOL™, Aspen ACX™ Upgrade to ACOL™, Aspen Adsim®, Aspen Advisor™, Aspen Aerotran®, Aspen Alarm & Event™, Aspen APLE™, Aspen Apollo™, Aspen AtOMS™, Aspen Batch and Event Extractor, Aspen Batch Plus®, Aspen Batch.21™, Aspen Batch.21™ CBT, Aspen BatchCAD™, Aspen BatchSep™, Aspen Blend Model Library™, Aspen Blend™, Aspen BP Crude Oil Database, Aspen Calc CBT, Aspen Calc™, Aspen Capable-to-Promise®, Aspen CatRef®, Aspen Chromatography®, Aspen Cim-IO Core™, Aspen Cim-IO™ for @AGlance, Aspen Cim-IO™ for ABB 1180/1190 via DIU, Aspen Cim-IO™ for Bailey SemAPI, Aspen Cim-IO™ for DDE, Aspen Cim-IO™ for Eurotherm Gauge via DCP, Aspen Cim-IO™ for Fisher-Rosemount Chip, Aspen Cim-IO™ for Fisher-Rosemount RNI, Aspen Cim-IO™ for Foxboro FOXAPI, Aspen Cim-IO™ for G2, Aspen Cim-IO™ for GE FANUC via HCT, Aspen Cim-IO™ for Hitachi Ex Series, Aspen Cim-IO™ for Honeywell TDC 3000 via HTL/access, Aspen Cim-IO™ for Intellution Fix, Aspen Cim-IO™ for Measurex MCN, Aspen Cim-IO™ for Measurex ODX, Aspen Cim-IO™ for Moore Apacs via Nim (RNI), Aspen Cim-IO™ for OPC, Aspen Cim-IO™ for PI, Aspen Cim-IO™ for RSLinx, Aspen Cim-IO™ for SetCim/InfoPlus-X/InfoPlus.21, Aspen Cim-IO™ for Toshiba Tosdic, Aspen Cim-IO™ for ULMA 3D, Aspen Cim-IO™ for Westinghouse, Aspen Cim-IO™ for WonderWare InTouch, Aspen Cim-IO™ for Yokogawa ACG10S, Aspen Cim-IO™ for Yokogawa EW3, Aspen Collaborative Forecasting™, Aspen Compliance.21™, Aspen COMThermo®, Aspen CPLEX Optimizer, Aspen CPLEX Optimizer for DPO, Aspen Crude Manager™, Aspen Crude Trading & Marketing™, Aspen Custom Modeler®, Aspen Data Source Architecture™, Aspen Decision Analyzer™, Aspen Demand Manager™, Aspen DISTIL™, Aspen Distribution Scheduler™, Aspen DMCplus®, Aspen DMCplus® CBT, Aspen DMCplus® Composite, Aspen Downtime Monitoring Application, Aspen DPO™, Aspen Dynamics®, Aspen eBRS™, Aspen FCC®, Aspen FIHR™, Aspen FLARENET™, Aspen Fleet Operations Management™, Aspen FRAN™, Aspen Fuel Gas Optimizer™, Aspen Grade-IT™, Aspen Harwell Subroutine Library™, Aspen Hetran®, Aspen HPI Library, Aspen HTFS Research Network™, Aspen HX-Net Operations™, Aspen HX-Net®, Aspen Hydrocracker®, Aspen Hydrotreater™, Aspen HYSYS Amines™, Aspen HYSYS Crude™, Aspen HYSYS Data Rec™, Aspen HYSYS Dynamics™, Aspen HYSYS Johnson Matthey Reactor Models™, Aspen HYSYS OLGAS 3-Phase™, Aspen HYSYS OLGAS™, Aspen HYSYS OLI Interface™, Aspen HYSYS Optimizer™, Aspen HYSYS PIPESYS™, Aspen HYSYS Tacite™, Aspen HYSYS Upstream Dynamics™, Aspen HYSYS Upstream™, Aspen HYSYS®, Aspen Icarus Process Evaluator®, Aspen Icarus Project Manager®, Aspen Icarus Project Scheduler™, Aspen InfoPlus.21®, Aspen Inventory Management & Operations Scheduling™, Aspen Inventory Planner™, Aspen IQmodel Powertools™, Aspen IQ™, Aspen Kbase®, Aspen Lab.21, Aspen MBO™, Aspen MPIMS™, Aspen Multivariate Server™, Aspen MUSE™, Aspen OnLine®, Aspen Open Simulation Environment Base™, Aspen Operations Manager - Event Management™, Aspen Operations Manager - Integration Infrastructure™, Aspen Operations Manager - Integration Infrastructure™ Advisor, Aspen Operations Manager - Integration Infrastructure™ Base, Aspen Operations Manager - Integration Infrastructure™ COM, Aspen Operations Manager - Integration Infrastructure™ Files, Aspen Operations Manager - Integration Infrastructure™ IP.21/SAP-PPPI, Aspen Operations Manager - Integration Infrastructure™ IP21, Aspen Operations Manager - Integration Infrastructure™ OPC, Aspen Operations Manager - Integration Infrastructure™ Orion, Aspen Operations Manager - Integration Infrastructure™ PIMS, Aspen Operations Manager - Integration Infrastructure™ Relational Databases, Aspen Operations Manager - Integration Infrastructure™ SAP R3, Aspen Operations Manager - Integration Infrastructure™ System Monitoring, Aspen Operations Manager - Integration Infrastructure™ Utilities, Aspen Operations Manager - Performance Scorecarding™, Aspen Operations Manager - Role Based Visualization™ MS SharePoint, Aspen Operations Manager - Role Based Visualization™ TIBCO, Aspen Operations Tracking™, Aspen Order Credit Management™, Aspen Orion Planning™, Aspen Orion XT™, Aspen OSE™ - Oil & Gas Adapter, Aspen OSE™ - Oil & Gas Optimizer, Aspen PEP Process Library™, Aspen PIMS Advanced Optimization™, Aspen PIMS CPLEX Optimizer, Aspen PIMS Distributed Processing™, Aspen PIMS Enterprise Edition™, Aspen PIMS Global Optimization™, Aspen PIMS Mixed Integer Programming™, Aspen PIMS Simulator Interface™, Aspen PIMS Solution Ranging™, Aspen PIMS Submodel Calculator™, Aspen PIMS XNLP Optimizer™, Aspen PIMS XPRESS Optimizer, Aspen PIMS-SX, Aspen PIMS™, Aspen PIMSXCHG, Aspen PIPE™, Aspen Plant Planner & Scheduler™, Aspen Plant Scheduler Lite™, Aspen Plant Scheduler™, Aspen Plus HTRI Interface, Aspen Plus OLI Interface™, Aspen Plus Optimizer™, Aspen Plus SPYRO Equation Oriented Interface, Aspen Plus®, Aspen Plus® CBT, Aspen Polymers Plus®, Aspen PPIMS™, Aspen Process Explorer™, Aspen Process Explorer™ CBT, Aspen Process Manual™ Applied Rheology, Aspen Process Manual™ Bulk Solids Handling, Aspen Process Manual™ Crystallization, Aspen Process Manual™ Drying, Aspen Process Manual™ Gas Cleaning, Aspen Process Manual™ Internet Mode, Aspen Process Manual™ Intranet Mode, Aspen Process Manual™ Mini-Manuals, Aspen Process Manual™ Slurry Handling, Aspen Process Manual™ Solid Liquid Separation, Aspen Process Manual™ Solvent Extraction, Aspen Process Manual™ Waste Water Treatment, Aspen Process Order™, Aspen Process Recipe®, Aspen Process Tools™, Aspen Product Tracking, Aspen Production Control Web Server™, Aspen ProFES® 2P Wax, Aspen ProFES® Tranflo, Aspen Profile.21™, Aspen Properties®, Aspen Pumper Log™, Aspen Q Server™, Aspen Quality Management™, Aspen RateSep™, Aspen RefSYS CatCracker™, Aspen RefSYS Hydrocracker™, Aspen RefSYS Reformer™, Aspen RefSYS™, Aspen Report Writer™, Aspen Retail Automated Stock Replenishment™, Aspen Retail Resource Scheduling Optimization™, Aspen Richardson Cost Factor Manual™, Aspen Richardson General Construction Estimating Standards™, Aspen Richardson Process Plant Construction Estimating Standards™, Aspen Richardson WinRace Database™, Aspen RTO Watch™, Aspen SCM™, Aspen SmartStep Advanced™, Aspen Specialty Products Automated Stock Replenishment™, Aspen Specialty Products Resource Scheduling Optimization™, Aspen Split™, Aspen State Space Controller™, Aspen STX™ Upgrade to TASC™, Aspen SULSIM®, Aspen Supply Chain Analytics™ - Demand Management, Aspen Supply Chain Analytics™ - Plant Scheduling, Aspen Supply Chain Analytics™ - S&OP, Aspen Supply Chain Analytics™ - Supply Planning, Aspen Supply Chain Connect™, Aspen Supply Planner™, Aspen Supply Planning - Strategic Analyzer™, Aspen Tank Management™, Aspen TASC™, Aspen Teams®, Aspen TICP™, Aspen Transition Manager™, Aspen Utilities™, Aspen Voice Fulfillment Management™, Aspen Watch™, Aspen Water™, Aspen Web Fulfillment Management™, Aspen XPIMS™, Aspen XPRESS Optimizer, Aspen XPRESS Optimizer for DPO, Aspen Zyqad Development™, Aspen Zyqad™, aspenONE Product Trading & Blending™, SLM™, SLM Commute™, SLM Config Wizard™, the Aspen leaf logo, and Plantelligence are trademarks or registered trademarks of Aspen Technology, Inc., Cambridge, MA.

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

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

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

Corporate

Aspen Technology, Inc.
Ten Canal Park
Cambridge, MA 02141-2201
USA

Phone: (617) 949-1000
Toll Free: (1) (888) 996-7001
Fax: (617) 949-1030
Website http://www.aspentech.com

Technical Support

Online Technical Support Center

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

http://support.aspentech.com

You use the Online Technical Support Center to:

- Access current product documentation.
- Search for technical tips, solutions, and frequently asked questions (FAQs).
- Search for and download application examples.
- Search for and download service packs and product updates.
- Submit and track technical issues.
- Search for and review known limitations.
- Send suggestions.

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

- Technical advisories
- Product updates
- Service Pack announcements
- Product release announcements

Phone and E-mail

Customer support is also available by phone, fax, and e-mail for customers who have a current support contract for their product(s). Toll-free charges are listed where available; otherwise local and international rates apply.

For the most up-to-date phone listings, please see the Online Technical Support Center at:

http://support.aspentech.com

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

Table of Contents

1 Operations Overview

1.1 Engineering

As explained in the **HYSYS User Guide** and **HYSYS Simulation Basis** guide, HYSYS has been uniquely created with respect to the program architecture, interface design, engineering capabilities, and interactive operation. The integrated steady state and dynamic modeling capabilities, where the same model can be evaluated from either perspective with full sharing of process information, represent a significant advancement in the engineering software industry.

The various components that comprise HYSYS provide an extremely powerful approach to steady state process modeling. At a fundamental level, the comprehensive selection of operations and property methods allows you to model a wide range of processes with confidence. Perhaps even more important is how the HYSYS approach to modeling maximizes your return on simulation time through increased process understanding. The key to this is the Event Driven operation. By using a 'degrees of freedom' approach, calculations in HYSYS are performed automatically. HYSYS performs calculations as soon as unit operations and property packages have enough required information.

Any results, including passing partial information when a complete calculation cannot be performed, is propagated bi-directionally throughout the flowsheet. What this means is that you can start your simulation in any location using the available information to its greatest advantage. Since results are available immediately - as calculations are performed - you gain the greatest understanding of each individual aspect of your process.

The multi-flowsheet architecture of HYSYS is vital to this overall modeling approach. Although HYSYS has been designed to allow the use of multiple property packages and the creation of pre-built templates, the greatest advantage of using multiple flowsheets is that they provide an extremely effective way to organize large processes. By breaking flowsheets into smaller components, you can easily isolate any aspect for detailed analysis. Each of these sub-processes is part of the overall simulation, automatically calculating like any other operation.

The design of the HYSYS interface is consistent, if not integral, with this approach to modeling. Access to information is the most important aspect of successful modeling, with accuracy and capabilities accepted as fundamental requirements. Not only can you access whatever information you need when you need it, but the same information can be displayed simultaneously in a variety of locations. Just as there is no standardized way to build a model, there is no unique way to look at results. HYSYS uses a variety of methods to display process information - individual property views, the PFD, Workbook, Databook, graphical Performance Profiles, and Tabular Summaries. Not only are all of these display types simultaneously available, but through the object-oriented design, every piece of displayed information is automatically updated whenever conditions change.

The inherent flexibility of HYSYS allows for the use of third party design options and custom-built unit operations. These can be linked to HYSYS through OLE Extensibility.

This Engineering section covers the various unit operations, template and column subflowsheet models, optimization, utilities, and dynamics. Since HYSYS is an integrated steady state and dynamic modeling package, the steady state and dynamic modeling capabilities of each unit operation are described successively, thus illustrating how the information is shared between the two approaches. In addition to the Physical operations, there is a chapter for Logical operations, which are the operations that do not physically perform heat and material balance calculations, but rather, impart logical relationships between the elements that make up your process.

The following is a brief definition of categories used in this volume:

Term	Definition
Physical Operations	Governed by thermodynamics and mass/energy balances, as well as operation-specific relations.
Logical Operations	The Logical Operations presented in this volume are primarily used in Steady State mode to establish numerical relationships between variables. Examples include the Adjust and Recycle. There are, however, several operations such as the Spreadsheet and Set operation which can be used in Steady State and Dynamic mode.
Subflowsheets	You can define processes in a subflowsheet, which can then be inserted as a "unit operation" into any other flowsheet. You have full access to the operations normally available in the main flowsheet.
Columns	Unlike the other unit operations, the HYSYS Column is contained within a separate subflowsheet, which appears as a single operation in the main flowsheet.

Integrated into the steady state modeling is multi-variable optimization. Once you have reached a converged solution, you can construct virtually any objective function with the Optimizer. There are five available solution algorithms for both unconstrained and constrained optimization problems, with an automatic backup mechanism when the flowsheet moves into a region of non-convergence.

HYSYS offers an assortment of utilities which can be attached to process streams and unit operations. These tools interact with the process model and provide additional information.

In this guide, each operation is explained in its respective chapters for steady state and dynamic modeling. A separate guide has been devoted to the principles behind dynamic modeling. HYSYS is the first simulation package to offer dynamic flowsheet modeling backed up by rigorous property package calculations.

Refer to **Section 1.6 - HYSYS Dynamics** in the **HYSYS Dynamic Modeling** guide for more information.

The HYSYS Dynamics license is required to use the features in the HYSYS dynamics mode.

HYSYS has a number of unit operations, which can be used to assemble flowsheets. By connecting the proper unit operations and streams, you can model a wide variety of oil, gas, petrochemical, and chemical processes.

Included in the available operations are those which are governed by thermodynamics and mass/energy balances, such as Heat Exchangers, Separators, and Compressors, and the logical operations like the Adjust, Set, and Recycle. A number of operations are also included specifically for dynamic modeling, such as the Controller, Transfer Function Block, and Selector. The Spreadsheet is a powerful tool, which provides a link to nearly any flowsheet variable, allowing you to model “special” effects not otherwise available in HYSYS.

In modeling operations, HYSYS uses a Degrees of Freedom approach, which increases the flexibility with which solutions are obtained. For most operations, you are not constrained to provide information in a specific order, or even to provide a specific set of information. As you provide information to the operation, HYSYS calculates any unknowns that can be determined based on what you have entered.

For instance, consider the Pump operation. If you provide a fully-defined inlet stream to the pump, HYSYS immediately passes the composition and flow to the outlet. If you then provide a percent efficiency and pressure rise, the outlet and energy streams is fully defined. If, on the other hand, the flowrate of the inlet stream is undefined, HYSYS cannot calculate any outlet conditions until you provide three parameters, such as the efficiency, pressure rise, and work. In the case of the Pump operation, there are three degrees of freedom, thus, three parameters are required to fully define the outlet stream.

All information concerning a unit operation can be found on the tabs and pages of its property view. Each tab in the property view contains pages which pertain to the unit operation, such as its stream connections, physical parameters (for example, pressure drop and energy input), or dynamic parameters such as vessel rating and valve information.

1.2 Operations

1.2.1 Installing Operations

There are a number of ways to install unit operations into your flowsheet. The operations which are available depends on where you are currently working (main flowsheet, template subflowsheet or column subflowsheet). If you are in the main flowsheet or template environments, all operations are available, except those associated specifically with the column, such as reboilers and condensers. A smaller set of operations is available within the column subflowsheet.

For detailed information on installing unit operations, refer to:

- **Section 8.1 - Installing Objects**
- **Section 7.23.2 - Installing Streams or Operations**

in the **HYSYS User Guide**.

The two primary areas from which you can install operations are the UnitOps property view and the Object Palette.

The operations are divided into categories with each category containing a number of individual operations. For the main flowsheet, the available operations are categorized in the following table.

Operation Category	Types
All	• All Unit Operations
Vessels	• 3-Phase Separator • Continuous Stirred Tank Reactor • Conversion Reactor • Equilibrium Reactor • Gibbs Reactor • Reboiler • Separator • Tank
Heat Transfer Equipment	• Air Cooler • Cooler • Fired Heater • Heat Exchanger • Heater • LNG
Rotating Equipment	• Compressor • Expander • Pump

Operation Category	Types
Piping Equipment	• Aspen Hydraulics Sub-Flowsheet • Compressible Gas Pipe • Liquid-liquid Hydrocyclone • Mixer • Pipe Segment • PIPESIM • PIPESIM Enhanced Link • PIPESYS Extension • Relief Valve • Tee • Valve
Solids Handling	• Baghouse Filter • Cyclone • Hydrocyclone • Rotary Vacuum Filter • Simple Solid Separator
Reactors	• Continuous-Stirred Tank Reactor (CSTR) • Conversion Reactor • Equilibrium Reactor • Gibbs Reactor • Plug Flow Reactor (PFR) • SULSIM Extension
Prebuilt Columns	• 3 Stripper Crude • 4 Stripper Crude • Absorber • Distillation • FCCU Main Fractionator • Liquid-Liquid Extractor • Reboiled Absorber • Refluxed Absorber • Three Phase Distillation • Vacuum Resid Tower
Short Cut Columns	• Component Splitter • Shortcut Column
Sub-Flowsheets	• 3 Stripper Crude • 4 Stripper Crude • Absorber • Aspen Hydraulics Sub-Flowsheet • Column Sub-Flowsheet • Distillation • FCCU Main Fractionator • Liquid-Liquid Extractor • MASSBAL Sub-Flowsheet • Reboiled Absorber • Refluxed Absorber • Standard Sub-Flowsheet • Three Phase Distillation • Vacuum Resid Tower

For information on the RefSYS operations refer to the **RefSYS Option Guide**.

For information on the Upstream operations refer to the **Upstream Option Guide**.

Operation Category	Types
Logicals	• Adjust • Balance • Black Oil Translator • Boolean And • Boolean CountDown • Boolean CountUp • Boolean Latch • Boolean Not • Boolean OffDly • Boolean OnDly • Boolean Or • Boolean XOr • Cause And Effect Matrix • Digital Control Point • DMCplus Controller • External Data Linker • MPC Controller • Parametric Unit Operation • PID Controller • Ratio Controller • Recycle • Selector Block • Set • Split Range Controller • Spreadsheet • Stream Cutter • Surge Controller • Transfer Function Block
Extensions	• User Defined
User Ops	• User Defined
Electrolyte Equipment	• Neutralizer • Precipitation • Crystalizer
RefSYS Ops	• Fluidized Catalytic Cracking • Manipulator • Petroleum Distillation • Petroleum Feeder • Petroleum Yield Shift Reactor • Product Blender
Upstream Ops	• Delumper • Liquid-liquid Hydrocyclone • Lumper • PIPESIM

The electrolyte operations are only available if your case is an electrolyte system (the selected fluid package must support electrolyte).

Prior to describing each of the unit operations, a quick overview of the material and energy streams is provided, as they are the means of transferring process information between operations.

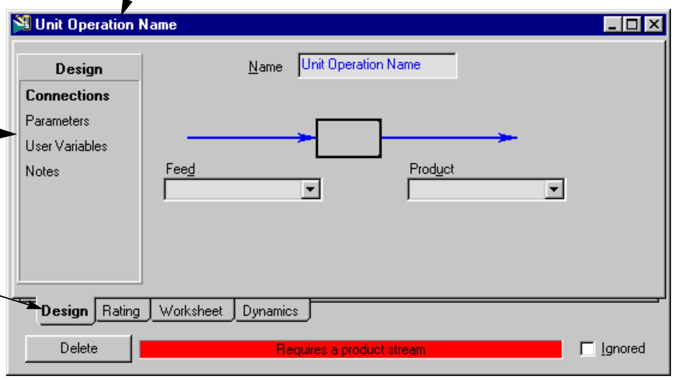
1.2.2 Unit Operation Property View

Although each unit operation differs in functionality and operation, in general, the unit operation property view remains fairly consistent in its overall appearance. The figure below shows a generic property view for a unit operation.

Figure 1.1

The various pages of the active tab.

The Name of the unit operation.



The active tab of the property view.

Most operation property view contains the following three common objects:

- **Delete** button. This button enables you to delete the unit operation from the current simulation case. Only the unit operation is deleted, any streams attached to the unit operation is left in the simulation case.
- **Status bar**. This bar displays messages associated to the calculation status of the unit operation. The messages also indicate the missing or incorrect data in the operation.
- **Ignore** checkbox. This checkbox enables you to toggle between including or excluding the unit operation in the simulation process calculation.

To ignore the operation during calculations, select the checkbox. HYSYS completely disregards the operation until you restore the operation to an active state by clearing the checkbox.

The Operation property view also contain several different tabs which are operation specific, however the Design, Ratings, Worksheet, and Dynamics tabs can usually be found in each unit operation property view and have similar functionality.

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

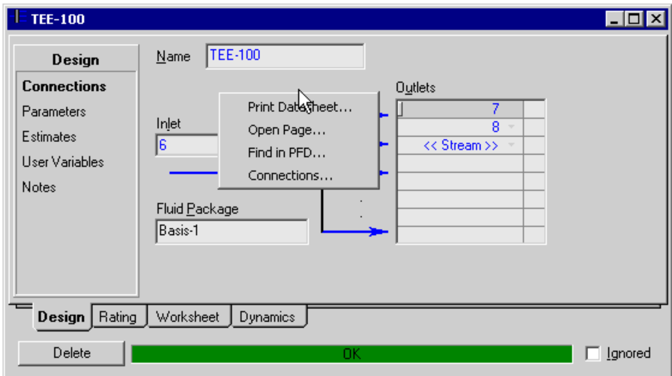
Tab	Description
Design	Connects the feed and outlet streams to the unit operation. Other parameters such as pressure drop, heat flow, and solving method are also specified on the various pages of this tab.
Ratings	Rates and Sizes the unit operation vessel. Specification of the tab is not always necessary in Steady State mode, however it can be used to calculate vessel hold up.
Worksheet	Displays the Conditions, Properties, Composition, and Pressure Flow values of the streams entering and exiting the unit operation.
Dynamics	Sets the dynamic parameters associated with the unit operation such as valve sizing and pressure flow relations. Not relevant to steady state modelling. For information on dynamic modelling implications of this tab, refer to the **HYSYS Dynamic Modeling** guide.

If negative pressure drop occurs in a vessel, the operation will not solve and a warning message appears in the status bar.

Object Inspect Menu

To access the Object Inspect menu of a unit operation property view, right-click on any empty area of the property view.

Figure 1.2



The unit operation property view all have the following common commands in the Object Inspect menu:

Refer to **Section 9.2.2 - Printing Datasheets** from the **HYSYS User Guide** for more information.

Command	Description
Print Datasheet	Enables you to access the Select DataBlocks to Print property view.
Open Page	Enables you to open the active page into a new property view.
Find in PFD	Enables you to locate and display the object icon in the PFD property view. This command is useful if you already have access to an object's property view and want to see where the object is located in the PFD. This command is only available in the Object Inspect menu of the HYSYS stream & operation property views.
Connections	Enables you to access the **Logical Connections For... Property View**.

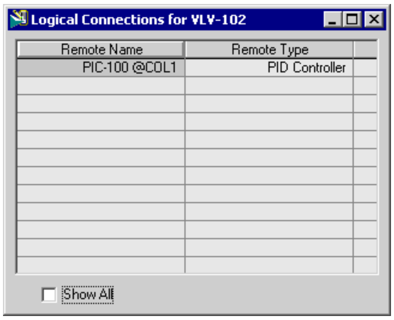
Logical Connections For... Property View

The Logical Connections for... property view enables you to determine simulation dependencies between objects which are not otherwise shown via connecting lines on the PFD. Certain

HYSYS operations can **write** to any other object and if the user is looking at the object being written to, they have no way of telling this, other than that the value might be changing. For example, one can determine if one spreadsheet is **writing** to another.

The Logical Connections for... property view is different if accessed from a Spreadsheet property view since there is an additional column (This Name) in the table. The This Name column displays the spreadsheet cell that contains the information/variable connected to the spreadsheet.

Figure 1.3



The table in the Logical Connections for... property view contains the following columns:

- **Remote Name** column displays the name of the operation or stream being written to or read from the active object.

  Double-click on a particular entry of the **Remote Name** column to open the property view of the operation or stream.
- **Remote Type** column displays the operation type (pump, valve, stream, and so forth) of the remote object from the current/active property view.

The **Show All** checkbox enables you toggle between displaying or hiding all the other operations and streams that the selected object knows about. Duplicate connectivity information may be shown otherwise (either via a line on the PFD or some place else in a Logical operations property view, for example). Usually, you do not need to select this checkbox.

There is only one Show All checkbox for your HYSYS session. When the checkbox is changed, the current setting is effective for all Logical Connections For... property view.

To access the Logical Connections for... view of a HYSYS PFD object:

1. Open the object's property view.
2. Right-click in an empty area of the object's property view. The Object Inspect menu associated to the object appears.
3. Select **Connections** command from the Object Inspect menu.

The information displayed in the Logical Connections for... property view is primarily use for the Spreadsheet, Cause and Effect Matrix operation, Event Scheduler operation, and any other operations that read / write from / to these property views.

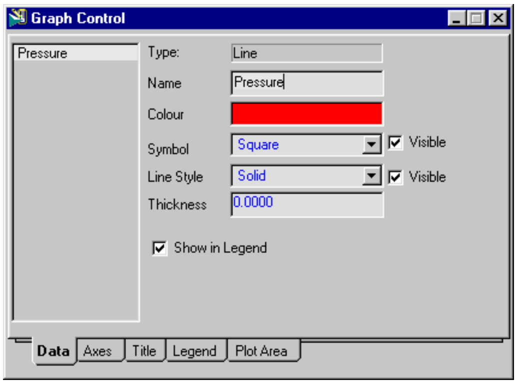
1.3 Common Property Views

Each operation in HYSYS contains some common information and options. These information and options are grouped into common property views, tabs, and pages. The following sections describe the common objects in HYSYS operation property view.

1.3.1 Graph Control Property View

The Graph Control property view and its options are available for all plots in HYSYS.

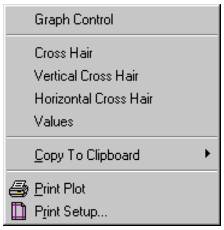
Figure 1.4



Refer to **Section 10.4 - Graph Control** in the **HYSYS User Guide** for more information.

The options are grouped into five tabs:

- Data. Contains options that enable you to modify the variable characteristics (type, name, colour, symbol, line style, and line thickness) of the plot.
- Axes. Contains options that enable you to modify the axes characteristics (label name, display format, and axes value range) of the plot.
- Title. Contains options that enable you to modify the title characteristics (label, font style, font colour, borders, and background colour) of the plot.
- Legend. Contains options that enable you to modify the legend characteristics (border, background colour, font style, font colour, and alignment) of the plot.
- Plot Area. Contains options that enable you to modify the plot characteristics (background colour, grid colour, frame colour, and cross hair colour) of the plot.



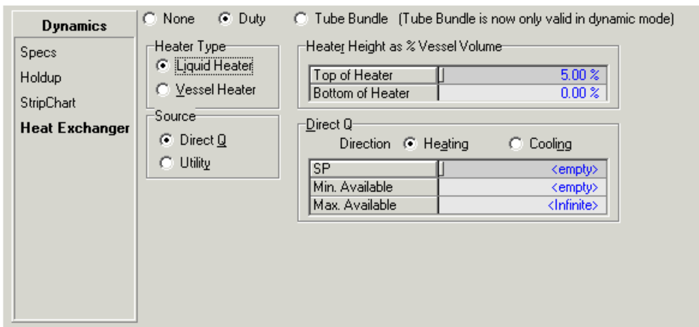
To access the Graph Control property view, do one of the following:

- Right-click any spot on an active plot and select the **Graph Control** command from the Object Inspect menu.
- Click in the plot area to make the plot the active object. Then, either double-click on the plot Title or Legend to access the respective tab of the Graph Control property view.

1.3.2 Heat Exchanger Page

The Heat Exchanger page in the Dynamics tab for most vessel unit operations in HYSYS contains the options use to configure heat transfer method within the unit operation.

Figure 1.5



There are three options to choose from:

- **None** radio button option indicates that there is no energy stream or heat exchanger in the vessel. The Heat Exchanger page is blank and you do not have to specify an energy stream for the unit operation to solve.
- **Duty** radio button option indicates that there is an energy stream in the vessel. The Heat Exchanger page contains the HYSYS standard heater or cooler parameters and you have to specify an energy stream for the unit operation to solve.
- **Tube Bundle** radio button option indicates that there is heat exchanger in the vessel and enables you to simulate a kettle reboiler or chiller. The Heat Exchanger page

contains the parameters used to configure a heat exchanger and you have to specify material streams of the heat exchanger for the unit operation to solve.

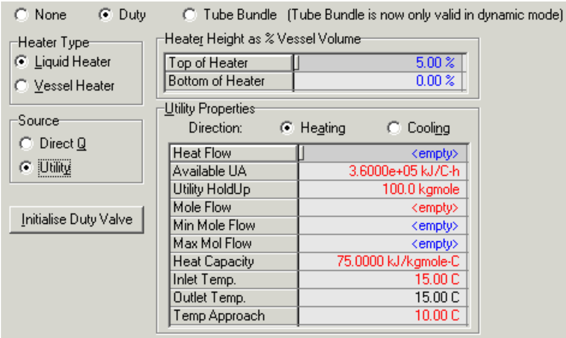
The Tube Bundle option is only available in Dynamics mode.

The Tube Bundle option is only available for the following unit operations: Separator, Three Phase Separator, Condenser, and Reboiler.

Duty Radio Button

When you select the **Duty** radio button the following options are available.

Figure 1.6



Heater Type Group

In the Heater Type group, there are two heating methods available to the general vessel operation:

- Vessel Heater
- Liquid Heater

If you select the Vessel Heater radio button, 100% of the duty specified or calculated in the **SP** field is applied to the vessel's holdup.

$$Q = Q_{Total} \tag{1.1}$$

where:

Q = *total heat applied to the holdup*

Q_{Total} = *duty calculated from the duty source*

If you select the Liquid Heater radio button, the duty applied to the vessel depends on the liquid level in the tank. You must specify the heater height in the **Top of Heater** and **Bottom of Heater** cells that appear with Heater Height as % Vessel Volume group.

The heater height is expressed as a percentage of the liquid level in the vessel operation. The default values are 5% for the *Top of the Heater* and 0% for the *Bottom of the Heater*. These values are used to scale the amount of duty that is applied to the vessel contents.

$$\begin{aligned} Q &= 0 && (L < B) \\ Q &= \frac{L-B}{T-B}Q_{Total} && (B \le L \le T) \\ Q &= Q_{Total} && (L > T) \end{aligned} \tag{1.2}$$

where:

L = *liquid percent level (%)*

T = *top of heater (%)*

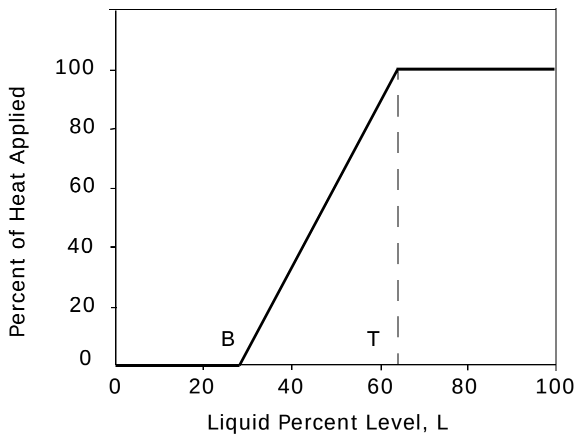
B = *bottom of heater (%)*

The Percent Heat Applied can be calculated as follows:

$$Percent\ Heat\ Applied = \frac{Q}{Q_{Total}} \times 100\% \tag{1.3}$$

It is shown that the percent of heat applied to the vessel's holdup directly varies with the surface area of liquid contacting the heater.

Figure 1.7



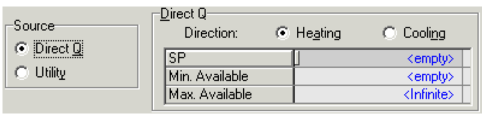
Percent Heat Applied for a Liquid Heater

Duty Source/Source Group

In the Duty Source/Source group, you can choose whether HYSYS calculates the duty applied to the vessel from a direct energy source or from a utility source.

- If you select the Direct Q radio button, the Direct Q group appears, and you can directly specify the duty applied to the holdup in the **SP** field.

Figure 1.8

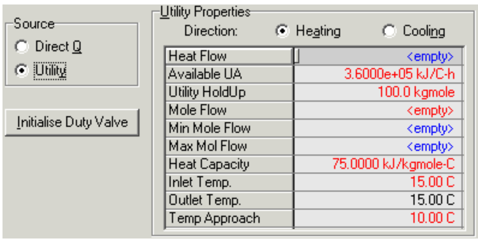


The following table describes the purpose of each object in the Direct Q group.

Object	Description
SP	The heat flow value in this cell is the same value specified in the Duty field of the Parameters page on the Design tab. Any changes made in this cell is reflected on the Duty field of the Parameters page on the Design tab.
Min. Available	Allows you to specify the minimum amount of heat flow.
Max. Available	Allows you to specify the maximum amount of heat flow.

- If you select the Utility radio button, the Utility Properties group appears, and you can specify the flow of the utility fluid.

Figure 1.9



For more information regarding how the utility option calculates duty, refer to **Chapter 5 - Logical Operations**.

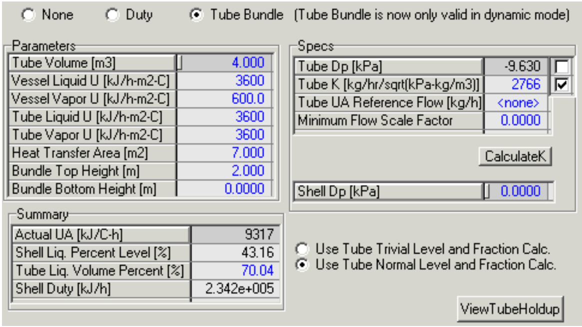
The duty is then calculated using the local overall heat transfer coefficient, the inlet fluid conditions, and the process conditions. The calculated duty is then displayed in the **SP** field or the **Heat Flow** field.

If you select the **Heating** radio button, the duty shown in the SP field or Heat Flow field is added to the holdup. If you select the **Cooling** radio button, the duty shown in the SP field or Heat Flow field is subtracted from the holdup.

Tube Bundle Radio Button

When you select the Tube Bundle radio button, the following options are available.

Figure 1.10



The Tube Bundle option is only available in Dynamics mode.

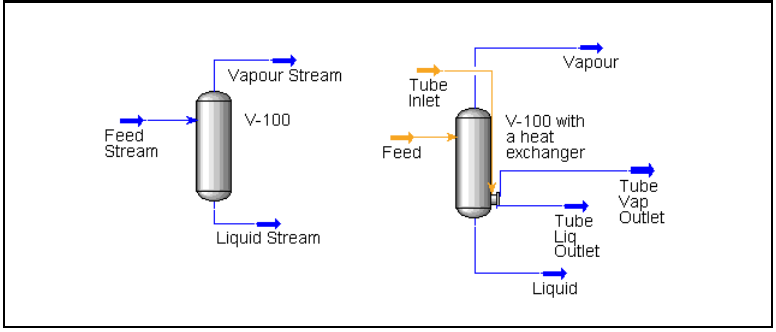
If you had an energy stream attached to the unit operation, HYSYS automatically disconnects the energy stream when you switch to the Tube Bundle option.

The Tube Bundle option allows you to configure a shell tube heat exchanger (for example, kettle reboiler or kettle chiller).

- In the kettle reboiler, the process fluid is typically on the shell side and the process fluid is fed into a liquid "pool" which is heated by a number of tubes. A weir limits the amount of liquid in the pool. The liquid overflow is placed under level control and provides the main liquid product. The vapor is circulated back to the vessel.
- In the kettle chiller, the process fluid is typically on the tube side with a refrigerant on the shell side. The refrigerant if typically pure and cools by evaporation. The setup is similar to the reboiler except that there is no weir or level control.

The unit operation icon in the PFD also changes to indicate that a heat exchanger has been attached to the unit operation.

Figure 1.11



The following table lists and describes the options available to configure the heat exchanger:

Object	Description
Parameters group	
Tube Volume cell	Allows you to specify the volume of the tubes in the heat exchanger.
Vessel Liquid U cell	Allows you to specify the heat transfer rate of the liquid in the shell.
Vessel Vapor U cell	Allows you to specify the heat transfer rate of the vapour in the shell.
Tube Liquid U cell	Allows you to specify the heat transfer rate of the liquid in the tube.
Tube Vapor U cell	Allows you to specify the heat transfer rate of the vapour in the tube.
Heat Transfer Area cell	Allows you to specify the total heat transfer area between the fluid in the shell and the fluid in the tube.
Bundle Top Height cell	Allows you to specify the location of the top tube/ bundle based on the height from the bottom of the shell.
Bundle Bottom Height cell	Allows you to specify the location of the bottom tube/ bundle based on the height from the bottom of the shell.
Specs group	
Tube Dp cell	Allows you to specify the pressure drop within the tubes. You have to select the associate checkbox in order to specify the pressure drop.
Tube K cell	Allows you to specify the pressure flow relationship value within the tubes. You have to select the associate checkbox in order to specify the pressure flow relationship value.

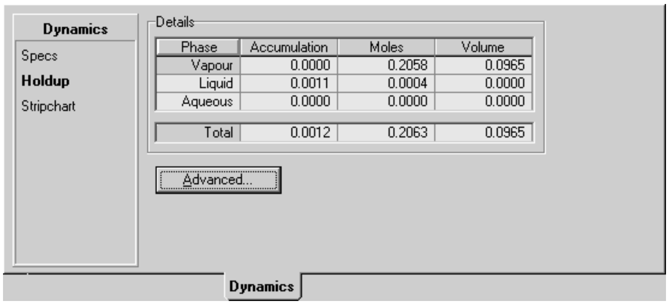
Object	Description
Tube UA Reference Flow cell	Allows you to set a reference point that uses HYSYS to calculate a more realistic UA value. If no reference point is set then UA is fixed. UA is the product of overall heat transfer multiply with overall heat transfer area, and depends on the flow rate. If a value is specified for the Reference Flow, the heat transfer coefficient is proportional to the $(\text{mass flow ratio})^{0.8}$. The equation below is used to determine the actual UA: $$UA_{\text{actual}} = UA_{\text{specified}} \times \left(\frac{\text{mass flow}_{\text{current}}}{\text{mass flow}_{\text{reference}}}\right)^{0.8}$$ Reference flows generally help to stabilize the system when you do shut downs and startups as well.
Minimum Flow Scale Factor cell	The ratio of mass flow at time *t* to reference mass flow is also known as flow scaled factor. The minimum flow scaled factor is the lowest value which the ratio is anticipated at low flow regions. This value can be expressed in a positive value or negative value. • A positive value ensures that some heat transfer still takes place at very low flows. • A negative value ignores heat transfer at very low flows. A negative minimum flow scale factor is often used in shut downs if you are not interested in the results or run into problems shutting down the heat exchanger. If the Minimum Flow Scale Factor is specified, the actual UA is calculated using the $\left(\frac{\text{mass flow}_{\text{current}}}{\text{mass flow}_{\text{reference}}}\right)^{0.8}$ ratio if the ratio is greater than the Min Flow Scale Factor. Otherwise the Min Flow Scale Factor is used.
Calculate K button	Allows you to calculate the K value based on the heat exchanger specifications.
Shell Dp cell	Allows you to specify the pressure drop within the shell.
Summary group	
Actual UA cell	Displays the calculated UA in Dynamics mode.
Shell Liq. Percent Level cell	Displays the calculated liquid level in the shell at percentage value.
Tube Liq. Volume Percent cell	Allows you to specify in percentage value the volume of liquid in the tube.
Shell Duty cell	Displays the calculated duty value in the shell.
Use Tube Trivial Level and Fraction Calc. radio button	Allows you to select the volume percent level variable for the vessel fraction calculation. This option uses a variable that is independent of the vessel shape or orientation.

Object	Description
Use Tube Normal Level and Fraction Calc. radio button	Allows you to select the liquid percent level variable for the vessel fraction calculation. This option uses a variable that is dependant of the vessel shape and orientation.
ViewTubeHoldUp button	Allows you to access the tube **HoldUp Property View**.

1.3.3 Holdup Page

Each unit operation in HYSYS has the capacity to store material and energy. The Holdup page contains information regarding the properties, composition, and amount of the holdup.

Figure 1.12



Most Holdup page contains the following common objects/ options:

Objects	Description
Phase column	Displays the phase of the fluid available in the unit operation's holdup volume. Each available phase occupies a volume space within the unit operation.
Accumulation column	Displays the rate of change of material in the holdup for each phase.
Moles column	Displays the amount of material in the holdup for each phase.
Volume column	Displays the holdup volume of each phase.

Objects	Description
Total row	Displays the sum of the holdup accumulation rate, mole value, and volume value.
Advanced button	Enables you to access the unit operation's **HoldUp Property View** that provides more detailed information about the holdup of that unit operation.

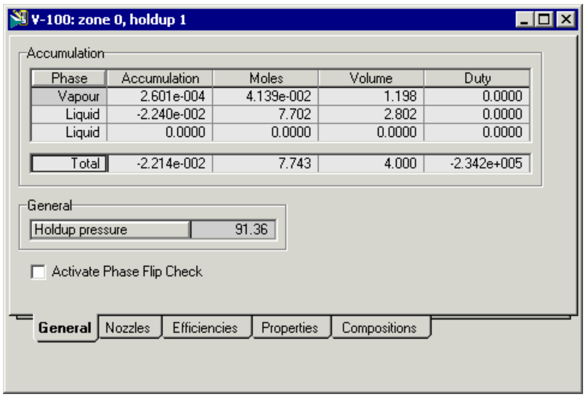
1.3.4 HoldUp Property View

Refer to **Section 1.3.7 - Advanced Holdup Properties** in the **HYSYS Dynamic Modeling** guide for more information.

The HoldUp property view displays the detailed calculated results of the holdup data in the following tabs:

- General. Displays the phase, accumulation, moles, volume, duty and holdup pressure of the heat exchanger.

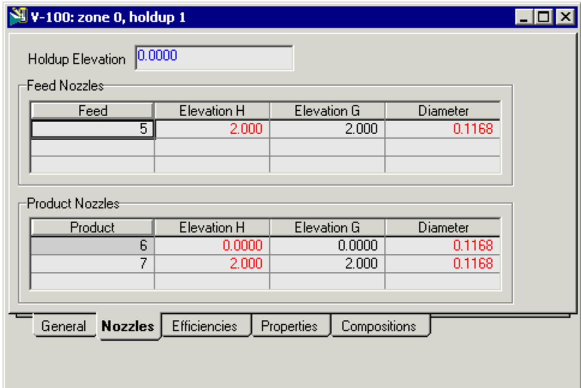
Figure 1.13



Select the **Active Phase Flip Check** checkbox to enable HYSYS to check if there is a phase flip between Liquid 1 (light liquid) and Liquid 2 (heavy liquid) during simulation and generate a warning message whenever the phase flip occur. If the checkbox is clear, HYSYS generates a warning only on the first time the phase flip occur.

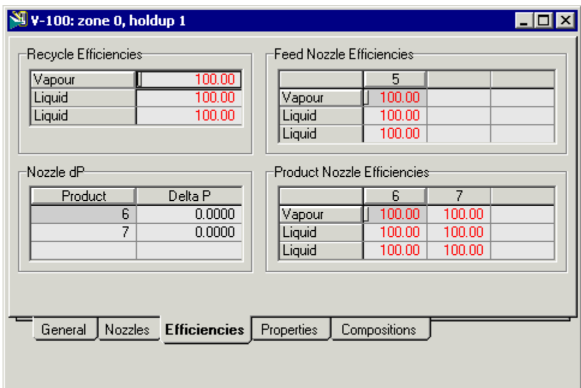
- Nozzles. Allows you to modify nozzle configuration attached to the heat exchanger.

Figure 1.14



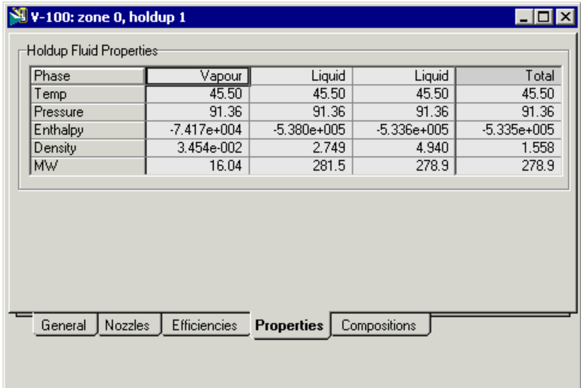
- Efficiencies. Allows you to modify the efficiency of the recycle, feed nozzle, and product nozzle of the heat exchanger.

Figure 1.15



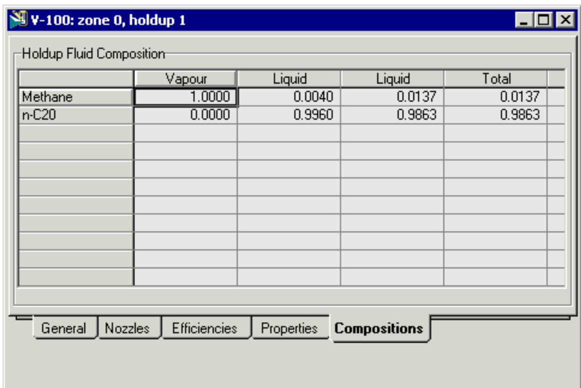
- Properties. Displays the temperature, pressure, enthalpy, density, and molecular weight of the holdup in the heat exchanger.

Figure 1.16



- Compositions. Displays the composition of the holdup in the heat exchanger.

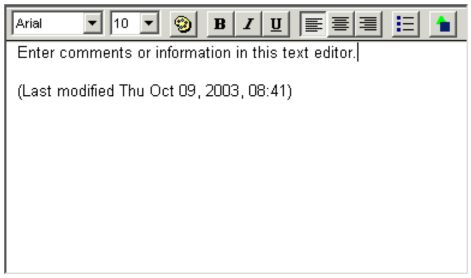
Figure 1.17



1.3.5 Notes Page/Tab

The Notes page/tab provides a text editor where you can record any comments or information regarding the specific unit operation or the simulation case in general.

Figure 1.18



To add a comment or information in the Notes page/tab:

1. Go to the **Notes** page/tab.
2. Use the options in the text editor toolbar to manipulate the appearance of the notes.

   The following table lists and describes the options available in the text editor toolbar.

Object	Icon	Description
Font Type		Use the drop-down list to select the text type for the note.
Font Size		Use the drop-down list to select the text size for the note.
Font Colour		Click this icon to select the text colour for the note.
Bold		Click this icon to bold the text for the note.
Italics		Click this icon to italize the text for the note.
Underline		Click this icon to underline the text for the note.
Align Left		Click this icon to left justify the text for the note.
Centre		Click this icon to center justify the text for the note.
Align Right		Click this icon to right justify the text for the note.

Object	Icon	Description
Bullets		Click this icon to apply bullets to the text for the note.
Insert Object		Click this icon to insert an object (for example an image) in the note.

3. Click in the large text field and type your comments.

   The date and time when you last modified the information in the text field will appear below your comments.

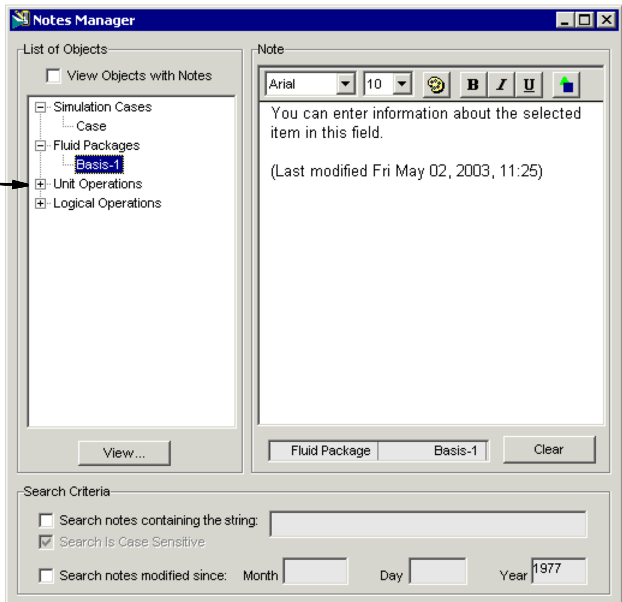
The information you enter in the Notes tab or page of any operations can also be viewed from the Notes Manager property view.

Notes Manager

The Notes Manager lets you search for and manage notes for a case. To access the Notes Manager, select **Notes Manager** command from the **Flowsheet** menu, or press the **CTRL G** hot key.

Figure 1.19

Click the **Plus** icon to expand the tree browser.



View/Add/Edit Notes

To view, add, or edit notes for an object, select the object in the List of Objects group. Existing object notes appear in the Note group.

- To add a note, type the text in the Note group. A time and date stamp appears automatically.
- To format note text, use the text tools in the Note group toolbar. You can also insert graphics and other objects.
- Click the **Clear** button to delete the entire note for the selected object.
- Click the **View** button to open the property view for the selected object.

Search Notes

The Notes Manager allows you to search notes in three ways:

- Select the **View Objects with Notes Only** checkbox (in the List of Objects group) to filter the list to show only objects that have notes.
- Select the **Search notes containing the string** checkbox, then type a search string. Only objects with notes containing that string appear in the object list.

  You can change the search option to be case sensitive by selecting the **Search is Case Sensitive** checkbox.

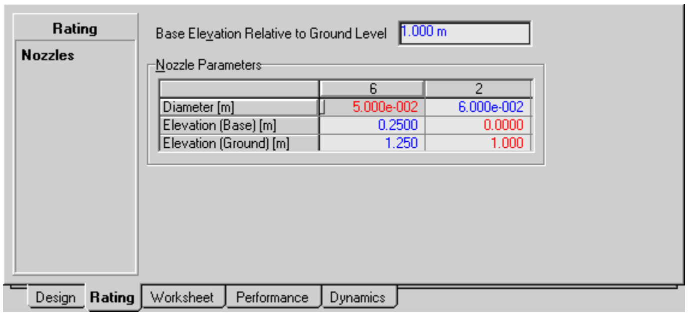
  The case sensitive search option is only available if you are searching by string.
- Select the **Search notes modified since** checkbox, then type a date. Only objects with notes modified after this date will appear in the object list.

1.3.6 Nozzles Page

The Nozzles page (from the Rating Tab) in most of the operations property view enables you to specify the elevation and diameter of the nozzles connected to the operation.

The Nozzles page is only available if the HYSYS Dynamics license is activated.

Figure 1.20



Refer to **Section 1.6.2 - Nozzles** in the **HYSYS Dynamic Modeling** guide for more information.

Depending on the type of operation, the options in the Nozzles page varies. The following table lists and describes the common options available in the page:

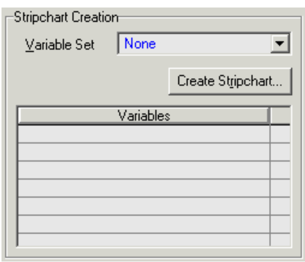
Object	Description
Base Elevation Relative to Ground Level field	Enables you to specify the height/elevation between the bottom of the operation and the ground.
Diameter row	Enables you to specify the diameter of the nozzle for each material stream flowing into and out of the operation.
Elevation (Base) row	Enables you to specify the height/elevation between the nozzle and the base of the operation.
Elevation (Ground) row	Enables you to specify the height/elevation between the nozzle and the ground.

1.3.7 Stripchart Page/Tab

Refer to **Section 11.7.3 - Strip Charts** in the **HYSYS User Guide** for more information about strip charts.

The Stripchart page or tab allows you to select and create a strip chart based on a default set of variable.

Figure 1.21

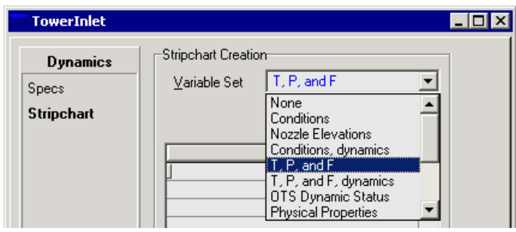


Depending on the object property view, the strip chart sets will contain variables appropriate for the object. For example, the strip chart set **ToolTip Properties** for a mixer will contain the following variables: Product Temperature, Product Pressure, and Product Molar Flow. The strip chart set **ToolTip Properties** for a separator will contain the following variables: Vessel Temperature, Vessel Pressure, and Liquid Volume Percent.

To select the strip chart set:

1. Open the object's property view, and access the **Stripchart** page or tab.
2. Select the strip chart set you want using the **Variable Set** drop-down list.

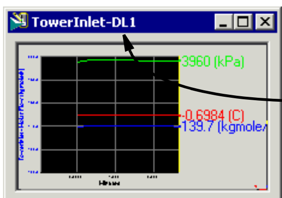
Figure 1.22



3. Clicking the **Create Stripchart** button.

   The new strip chart property view appears.

Figure 1.23



The new strip chart is automatically named:
objectname*-DL*n

where:

objectname *= name of the object*

n *= an integer number that increases each time a new strip chart with the same* ***objectname*** *is created*

Refer to **Section 11.7 - Databook** in the **HYSYS User Guide** for more information.

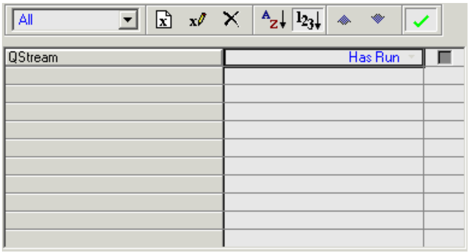
If you closed the strip chart property view, you can open the strip chart property view again using the options in the Databook property view.

1.3.8 User Variables Page/Tab

For more information on the user variables, refer to **Chapter 5 - User Variables** in the **HYSYS Customization Guide**.

The User Variables page or tab enables you to create and implement variables in the HYSYS simulation case.

Figure 1.24



The following table outlines options in the user variables toolbar:

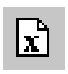
Object	Icon	Function
Current Variable Filter drop-down list		Enables you to filter the list of variables in the table based on the following types: • All • Real • Enumeration • Text • Code Only • Message
Create a New User Variable icon		Enables you to create a new user variable and access the Create a New User Variable property view.
Edit the Selected User Variable icon		Enables you to edit the configuration of an existing user variable in the table. You can also open the edit property view of a user variable by double-clicking on its name in the table.
Delete the Selected User Variable icon		Enables you to delete the select user variable in the table. HYSYS requires confirmation before proceeding with the deletion. If a password has been assigned to the User Variable, the password is requested before proceeding with the deletion.
Sort Alphabetically icon		Enables you to sort the user variable list in ascending alphabetical order.

Object	Icon	Function
Sort by Execution Order icon		Enables you to sort the user variable list according to the order by which they are executed by HYSYS. Sorting by execution order is important if your user variables have order dependencies in their macro code. Normally, you should try and avoid these types of dependencies.
Move Selected Variable Up In Execution Order icon		Enables you to move the selected user variable up in execution order.
Move Selected Variable Down In Execution Order icon		Enables you to move the selected user variable down in the execution order.
Show / Hide Variable Enabling Checkbox icon		Enables you to toggle between displaying or hiding the **Variable Enabling** checkboxes associated with each user variable. By default, the checkboxes are not displayed.

To add a user variable:

1. Access the **User Variables** page or tab in the object property view.
2. Click the **Create a New User Variable** icon.

   The Create New User Variable property view appears.

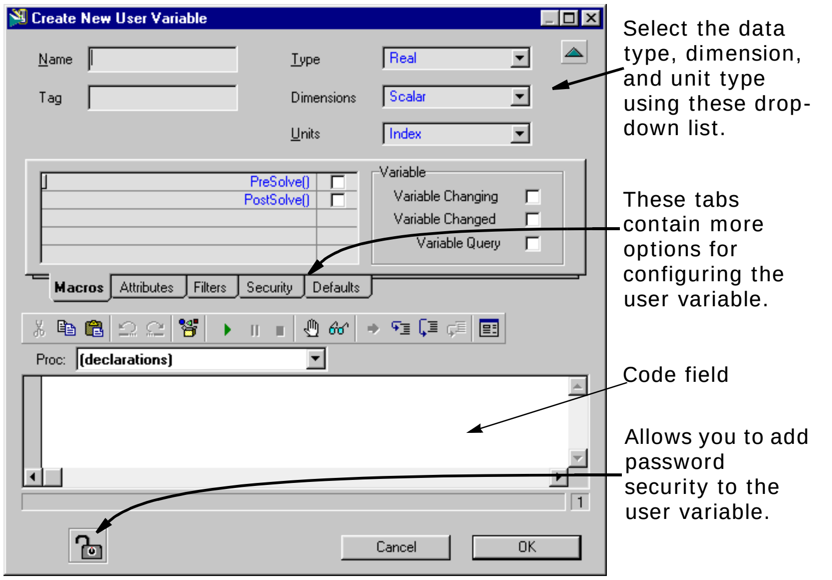


Create a New User Variable icon

3. In the **Name** field, type in the user variable name.

4. Fill in the rest of the user variable parameters as indicated by the figure below.

Figure 1.25



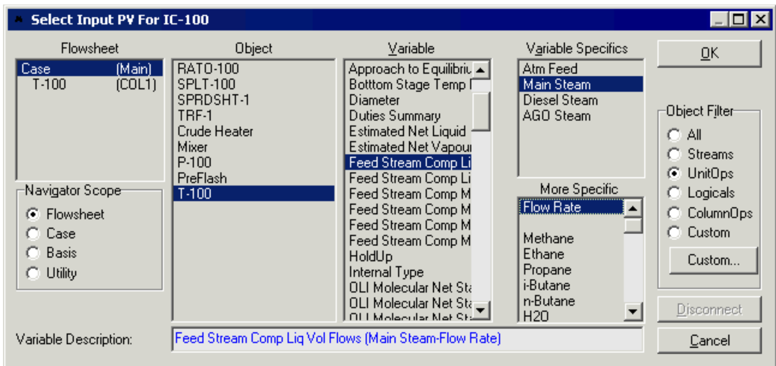
You can define your own filters on the Filters tab of the User Variable property view.

1.3.9 Variable Navigator Property View

Refer to **Section 11.21 - Variable Navigator** in the **HYSYS User Guide** for more information.

The Variable Navigator property view enables you to browse for and select variable, such as selecting a process variable for a controller or a strip chart.

Figure 1.26



Object	Description
Flowsheet / Case / Basis Object / Utility group	Enables you to select the flowsheet/case/basis object/ utility containing the variable you want. This type of objects available in this group depends on the selection in the Navigator Scope group.
Object group	Enables you to select the object containing the variable you want. The list of available objects depend on the flowsheet you selected in the Flowsheet group.
Variable group	Enables you to select the variable you want. The list of available variables depend on the object you selected in the Object group.
Variable Specifics group	Enables you to select a specific item of the variable. The list of available items depend on the variable you selected in the Variable group.
More Specific group	Enables you to select in detail the item of the variable you want. The list of available sub-items depend on the item you selected in the Variable Specifics group.
Navigator Scope group	Enables you to select the area/location containing the variable you want.

Object	Description
Variable Description field	Enables you to provide a name for the selected variable.
OK button	Enables you to confirm the selection of the variable and close the navigator property view. This button is only available when you have selected the variable in the groups.
Add button	Enables you to confirm the selection of the variable and keep the navigator property view open to select more variable. This button is only available when the operation allows multiple variable selection.
Object Filter group	Enables you to filter the types of objects displayed in the Object group.
Disconnect button	Enables you to remove/disconnect the selected variable and close the property view. This button is only available when a variable is selected in the navigator property view.
Close button	Enables you to close the navigator property view. This button is only available if you have selected multiple variable in the same navigator property view.
Cancel button	Enables you to close the navigator property view without making any changes or variable selection.

1.3.10 Worksheet Tab

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the unit operation.

- The Conditions and Composition pages contain selected information from the corresponding pages of the Worksheet tab for the stream property view.
- The Properties page displays the property correlations of the inlet and outlet streams of the unit operations. The following is a list of the property correlations:

- Vapour / Phase Fraction
- Temperature
- Pressure
- Actual Vol. Flow
- Mass Enthalpy
- Mass Entropy
- Molecular Weight
- Molar Density
- Vap. Frac. (molar basis)
- Vap. Frac. (mass basis)
- Vap. Frac. (volume basis)
- Molar Volume
- Act. Gas Flow
- Act. Liq. Flow
- Std. Liq. Flow
- Std. Gas Flow

- Mass Density
- Std. Ideal Liquid Mass Density
- Liquid Mass Density
- Molar Heat Capacity
- Mass Heat Capacity
- Thermal Conductivity
- Viscosity
- Surface Tension
- Specific Heat
- Z Factor
- Watson K
- Kinematic Viscosity
- Cp/Cv
- Lower Heating Value
- Mass Lower Heating Value
- Liquid Fraction
- Partial Pressure of CO2
- Avg. Liq. Density
- Heat of Vap.
- Mass Heat of Vap.

The Heat of Vapourisation for a stream in HYSYS is defined as the heat required to go from saturated liquid to saturated vapour.

- The PF Specs page contains a summary of the stream property view Dynamics tab.

The PF Specs page is relevant to dynamics cases only.

2 Column Operations

2.1 Column Subflowsheet

For detailed information about subflowsheet manipulation, refer to **Chapter 3 - Flowsheet** in the **HYSYS User Guide.**

The Column is a special type of subflowsheet in HYSYS. A subflowsheet contains equipment and streams, and exchanges information with the parent flowsheet through the connected internal and external streams. From the *main* simulation environment, the Column appears as a single, multi-feed multi-product operation. In many cases, you can treat the column in exactly that manner.

You can also work inside the Column subflowsheet. You can do this to “focus” your attention on the Column. When you move into the Column *build environment*, the main simulation is “cached.” All aspects of the main environment are paused until you exit the Column build environment. When you return to the Main Environment, the Desktop re-appears as it was when you left it.

You can also enter the Column build environment when you want to create a custom column configuration. Side equipment such as pump arounds, side strippers, and side rectifiers can be added from the Column property view in the main simulation. However, if you want to install multiple tray sections or multiple columns, you need to enter the Column build environment. Once inside, you can access the Column-specific operations (Tray Sections, Heaters/Coolers, Condensers, Reboilers, and sof forth) and build the column as you would any other flowsheet.

In this chapter, the use of the Column property view and Column Templates are explained. **Section 2.7 - Column-Specific Operations**, describes the unit operations available in the Column build environment.

If you want to create a custom column template for use in other simulations, on the File menu select the New command, and then select the Column sub-command. Since this is a column template, you can access the Column build environment directly from the Basis environment. Once you have created the template, you can store it on disk. Before you install the template in another simulation, ensure that the **Use Input Experts** checkbox in the Session Preferences property view is cleared.

Having a Column subflowsheet provides a number of advantages:

- isolation of the Column Solver.

- optional use of different Property Packages.
- construction of custom templates.
- ability to solve multiple towers simultaneously.

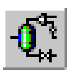
Isolation of the Column Solver

One advantage of the Column build environment is that it allows you to make changes, and focus on the Column without requiring a recalculation of the entire flowsheet. When you enter the Column build environment, HYSYS clears the Desktop by caching all property views that were open in the parent flowsheet. Then the property views that were open when you were last in the Column build environment are re-opened.

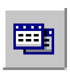
Once inside the Column build environment, you can access profiles, stage summaries, and other data, as well as make changes to Column specifications, parameters, equipment, efficiencies, or reactions. When you have made the necessary changes, simply run the Column to produce a new converged solution. The parent flowsheet cannot recalculate until you return to the parent build environment.

While in the Column subflowsheet, you can view the Workbook or PFD for both the Parent flowsheet or subflowsheet by using the Workbooks option or PFDs option in the Tools menu.

The subflowsheet environment permits easy access to all streams and operations associated with your column.



PFD icon



Workbook icon

- Click the **PFD** icon to view the column subflowsheet.
- If you want to access information regarding column product streams, click the **Workbook** icon to view the Column workbook, which displays the Column information exclusively.

Independent Fluid Package

HYSYS allows you to specify a unique fluid package for the Column subflowsheet. Here are some instances where a separate fluid package is useful:

- If a column does not use all of the components used in the main flowsheet, it is often advantageous to define a new fluid package with only the components that are necessary. This speeds up the column solution.
- In some cases, a different fluid package can be better suited to the column conditions. For example, if you want to redefine **Interaction Parameters** such that they are applicable for the operating range of the column.
- In Dynamic mode, different columns can operate at very different temperatures and pressures. With each fluid package, you can define a different dynamic model whose parameters can be regressed in the appropriate temperature and pressure range, thus, improving the accuracy and stability of the dynamic simulation.

Ability to construct Custom Column Configurations

Custom column configurations can be stored as templates, and recalled into another simulation. To create a custom template, on the File menu select the New command, and then select the Column sub-command. When you store the template, it has a *.col extension.

Complex custom columns and multiple columns can be simulated within a single subflowsheet using various combinations of subflowsheet equipment.

There exists a great deal of freedom when defining column configurations, and you can define column setups with varying degrees of complexity. You can use a wide array of column operations in a manner which is straightforward and flexible.

Column arrangements are created in the same way that you build the main flowsheet:

- accessing various operations.
- making the appropriate connections.
- defining the parameters.

Use of Simultaneous Solution Algorithm

The Column subflowsheet uses a simultaneous solver whereby all operations within the subflowsheet are solved simultaneously. The simultaneous solver permits you to install multiple unit operations within the subflowsheet (interconnected columns, for example) without the need for Recycle blocks.

Dynamic Mode

There are several major differences between the dynamic column operation and the steady state column operation. One of the main differences is the way in which the Column subflowsheet solves.

In steady state if you are in the Column subflowsheet, calculations in the main flowsheet are put on Hold until the focus is returned to the main flowsheet. When running in dynamics, calculations in the main flowsheet proceed at the same time as those in the Column subflowsheet.

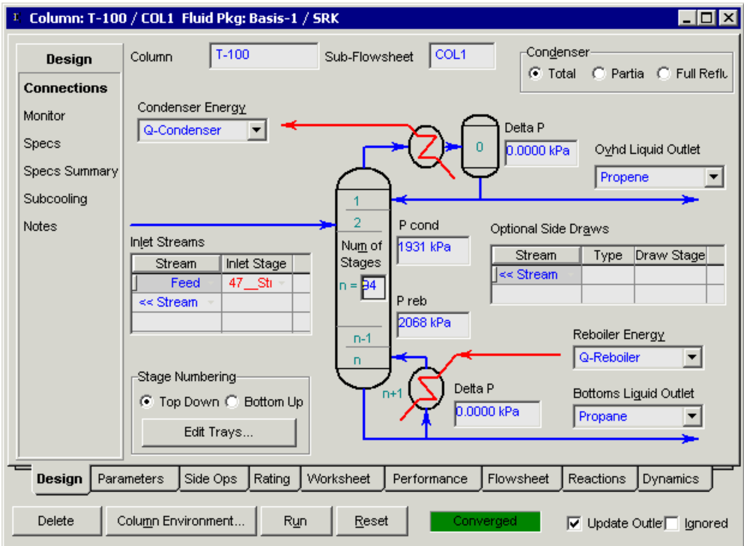
Another difference between the steady state column and the dynamic column is with the column specifications. Steady state column specifications are ignored in dynamics. To achieve the column specifications when using dynamics, control schemes must be added to the column.

Finally, although it is possible to turn off static head contributions for the rest of the simulation, this option does not apply to the column. When running a column in Dynamic mode, the static head contributions are always used in the column calculations.

Column Property View

The Column property view (the representation of the Column within the main or parent flowsheet) essentially provides you with complete access to the Column.

Figure 2.1



For more information, refer to **Section 2.4 - Column Property View**.

From the Column property view, you can change feed and product connections, specifications, parameters, pressures, estimates, efficiencies, reactions, side operations, and view the Profiles, Work Sheet, and Summary. You can also run the column from the main flowsheet just as you would from the Column subflowsheet.

Side equipment (for example, pump arounds and side strippers) is added from the Column property view.

If you want to make a minor change to a column operation (for instance, resize a condenser) you can call up that operation using the Object Navigator without entering the Column subflowsheet. Major changes, such as adding a second tray section, require you to enter the Column subflowsheet.

To access to the Column build environment, click the Column Environment button at the bottom of the Column property view.

Enter the Column subflowsheet to add new pieces of equipment, such as additional Tray Sections or Reboilers.

Main Flowsheet and Column Subflowsheet Relationship

Unlike other unit operations, the Column contains its own subflowsheet, which in turn, is contained in the Parent (usually the main) flowsheet. When you are working in the parent flowsheet, the Column appears just as any other unit operation, with multiple input and output streams, and various adjustable parameters.

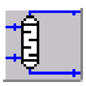
If you make a change to the Column while you are working in the parent, or main build environment, both the Column and the parent flowsheets are automatically recalculated.

When you install a Column, HYSYS creates a subflowsheet containing all operations and streams associated with the template you have chosen. This subflowsheet operates as a unit operation in the main flowsheet. **Figure 2.2** shows this concept of a Column subflowsheet within a main flowsheet.

Main Flowsheet / Subflowsheet Concept

Consider a simple absorber in which you want to remove CO2 from a gas stream using H2O as the solvent. A typical approach to setting up the problem would be as follows:

1. Create the gas feed stream, **FeedGas**, and the water solvent stream, **WaterIn**, in the main flowsheet.
2. Click the **Absorber** icon from the Object Palette.



Absorber icon

3. Specify the stream names, number of trays, pressures, estimates, and specifications. You must also specify the names of the outlet streams, **CleanGas** and **WaterOut.**
4. Run the Column from the main flowsheet Column property view.

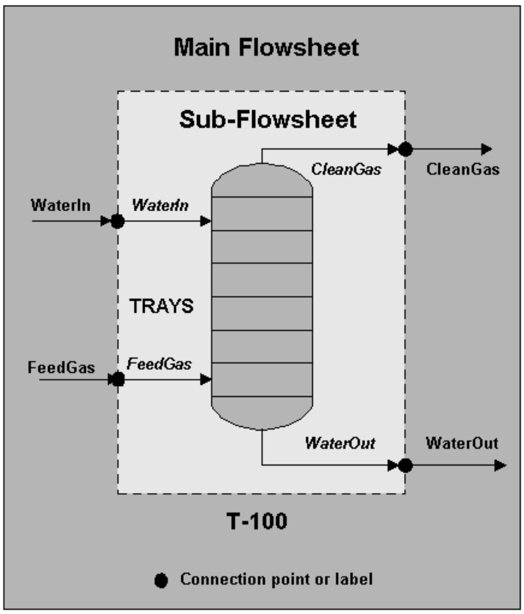
When you connected the streams to the tower, HYSYS created internal streams with the same names. The Connection Points or “Labels” serve to connect the main flowsheet streams to the subflowsheet streams and facilitate the information transfer between the two flowsheets.

A subflowsheet stream that is connected to a stream in the main flowsheet is automatically given the same name with “*@subflowsheet tag*” attached at the end of the name.

An example is the stream named “WaterIn” has the subflowsheet stream named “WaterIn@Col1”.

For instance, the main flowsheet stream WaterIn is **connected** to the subflowsheet stream WaterIn.

Figure 2.2



The connected streams do not necessarily have the same values. All specified values are identical, but calculated stream variables can be different depending on the fluid packages and transfer basis (defined on the Flowsheet tab).

When working in the main build environment, you “see” the Column just as any other unit operation, with a property view containing parameters such as the number of stages, and top and bottom pressures. If you change one of these parameters, the subflowsheet recalculates (just as if you had clicked the Run button); the main flowsheet also recalculates once a new column solution is reached.

However, if you are inside the Column subflowsheet build environment, you are working in an entirely different flowsheet. To make a major change to the Column such as adding a reboiler, you must enter the Column subflowsheet build environment. When you enter this environment, the main flowsheet is put on “hold” until you return.

If you delete any streams connected to the column in the main flowsheet, these streams are also deleted in the Column subflowsheet.

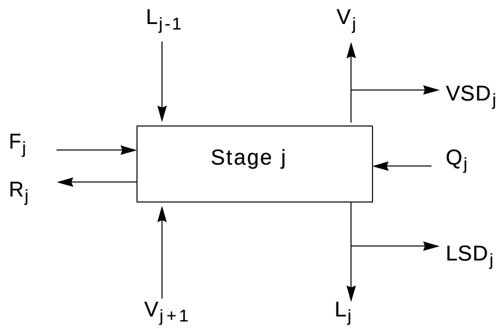
2.2 Column Theory

For information regarding the electrolyte column theory, refer to **Section 1.6.8 - HYSYS Column Operation** in the **HYSYS OLI Interface Reference Guide**.

Multi-stage fractionation towers, such as crude and vacuum distillation units, reboiled demethanizers, and extractive distillation columns, are the most complex unit operations that HYSYS simulates. Depending on the system being simulated, each of these towers consists of a series of equilibrium or non-equilibrium flash stages. The vapour leaving each stage flows to the stage above and the liquid from the stage flows to the stage below. A stage can have one or more feed streams flowing onto it, liquid or vapour products withdrawn from it, and can be heated or cooled with a side exchanger.

The following figure shows a typical stage j in a Column using the top-down stage numbering scheme. The stage above is j-1, while the stage below is j+1. The stream nomenclature is shown in the figure below.

Figure 2.3



F = Stage feed stream
L = Liquid stream travelling to stage below
V = Vapor stream travelling to stage above
LSD = Liquid side draw from stage
VSD = Vapor side draw from stage
Q = Energy stream entering stage

More complex towers can have pump arounds, which withdraw liquid from one stage of the tower and typically return it to a stage farther up the column. Small auxiliary towers, called sidestrippers, can be used on some towers to help purify side liquid products. With the exception of Crude distillation towers, very few columns have all of these items, but virtually any type of column can be simulated with the appropriate combination of features.

It is important to note that the Column operation by itself is capable of handling all the different fractionation applications. HYSYS has the capability to run cryogenic towers, high pressure TEG absorption systems, sour water strippers, lean oil absorbers, complex crude towers, highly non-ideal azeotropic distillation columns, and so forth. There are no programmed limits for the number of components and stages. The size of the column which you can solve depends on your hardware configuration and the amount of computer memory you have available.

The column is unique among the unit operations in the methods used for calculations. There are several additional underlying equations which are used in the column.

The Francis Weir equation is the starting point for calculating the liquid flowrate leaving a tray:

$$L_N = C\rho l_w h^{1.5} \tag{2.1}$$

where:

L_N = *liquid flowrate leaving tray N*

C = *units conversion constant*

ρ = *density of liquid on tray*

l_w = *weir length*

h = *height of liquid above weir*

The vapour flowrate leaving a tray is determined by the resistance equation:

$$F_{vap} = k\sqrt{\Delta P_{friction}} \tag{2.2}$$

where:

F_{vap} = *vapour flowrate leaving tray N*

k = *conductance, which is a constant representing the reciprocal of resistance to flow*

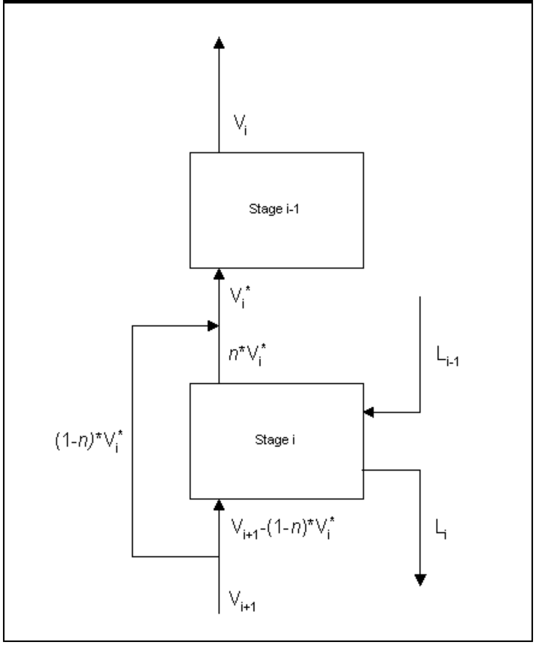
$\Delta P_{friction}$ = *dry hole pressure drop*

For columns the conductance, *k*, is proportional to the square of the column diameter.

The pressure drop across a stage is determined by summing the static head and the frictional losses.

It is possible to use column stage efficiencies when running a column in dynamics. The efficiency is equivalent to bypassing a portion of the vapour around the liquid phase, as shown in the figure below, where *n* is the specified efficiency.

Figure 2.4



HYSYS has the ability to model both weeping and flooding inside the column. If $\Delta P_{friction}$ is very small, the stage exhibits weeping. Therefore it is possible to have a liquid flow to the stage below even if the liquid height over the weir is zero.

For the flooding condition, the bulk liquid volume approaches the tray volume. This can be observed on the Holdup page in the Dynamics tab, of either the Column Runner or the Tray Section property view.

2.2.1 Three Phase Theory

For non-ideal systems with more than two components, boundaries can exist in the form of azeotropes, which a simple distillation system cannot cross. The formation of azeotropes in a three phase system provides a thermodynamic barrier to separating chemical mixtures.

Refer to **Section 2.3.2 - Templates** for further details on the three phase capabilities in HYSYS.

Distillation schemes for non-ideal systems are often difficult to converge without very accurate initial guesses. To aid in the initialization of towers, a Three Phase Input Expert is available to initialize temperatures, flows, and compositions.

For non-ideal multicomponent systems, DISTIL is an excellent tool for determining process viability. This conceptual design software application also determines the optimal feed tray location and allows direct export of column specifications to HYSYS for use as an initial estimate. Contact your local AspenTech representative for details.

2.2.2 Detection of Three Phases

Whenever your Column converges, HYSYS automatically performs a Three Phase Flash on the top stage. If a second liquid phase is detected, and no associated water draw is found, a warning message appears.

Look at the Trace Window for column convergence messages.

If there is a water draw, HYSYS checks the next stage for a second liquid phase, with the same results as above. This continues down the Tower until a stage is found that is two phase only.

If there is a three phase stage below a stage that was found to be two phase, the three phase stage is not detected because the checking would have ended in the previous two phase stage.

HYSYS always indicates the existence of the second liquid phase. This continues until the Column reverts to VLE operation, or all applicable stages have water draws placed on them.

2.2.3 Initial Estimates

Initial estimates are optional values that you provide to help the HYSYS algorithm converge to a solution. The better your estimates, the quicker HYSYS converges.

There are three ways for you to provide the column with initial estimates:

- Provide the estimate values when you first build the column.
- Go to the Profiles or Estimates page on the Parameters tab to provide the estimate values.
- Go to the Monitor or Specs page on the Design tab to provide values for the default specifications or add your own specifications.

Refer to **Section 2.3.2 - Templates** for more information regarding default specifications.

It is important to remember, when the column starts to solve for the first time or after the column has been reset, the specification values are also initial estimates. So if you replaced one of the original default specifications (overhead vapour flow, side liquid draw or reflux ratio) with a new active specification, the new specification value is used as initial estimates. For this reason it is recommended you provide reasonable specification values initially even if you can replace them while the column is solving or after the column has solved.

Although HYSYS does not require any estimates to converge to a solution, reasonable estimates help in the convergence process.

Temperatures

Temperature estimates can be given for any stage in the column, including the condenser and reboiler, using the Profiles page in the Parameters tab of the Column property view. Intermediate temperatures are estimated by linear interpolation. When large temperature changes occur across the condenser or bottom reboiler, it would be helpful to provide an estimate for the top and bottom trays in the tray section.

If the overhead product is a subcooled liquid, it is best to specify an estimated bubble-point temperature for the condenser rather than the subcooled temperature.

Mixing Rules at Feed Stages

When a feed stream is introduced onto a stage of the column, the following sequence is employed to establish the resulting internal product streams:

1. The entire component flow (liquid and vapour phase) of the feed stream is added to the component flows of the internal vapour and liquid phases entering the stage.
2. The total enthalpy (vapour and liquid phases) of the feed stream is added to the enthalpies of the internal vapour and liquid streams entering the stage.
3. HYSYS flashes the combined mixture based on the total enthalpy at the stage Pressure. The results of this process produce the conditions and composition of the vapour and liquid phases leaving the stage.

In most physical situations, the vapour phase of a feed stream does not come in close contact with the liquid on its feed stage. However if this is the case, the column allows you to split all material inlet streams into their phase components before being fed to the column. The **Split Inlets** checkbox can be selected in the **Setup** page of the **Flowsheet** tab. You can also set all the feed streams to a column to always split, by selecting the appropriate checkbox in the **Options** page from the **Simulation** tab of the Session Preferences property view.

Basic Column Parameters

Regardless of the type of column, the Basic Column Parameters remain at their input values during convergence.

Pressure

The pressure profile in a Column Tray Section is calculated using your specifications. You can either explicitly enter all stage pressures or enter the top and bottom tray pressures (and any intermediate pressures) such that HYSYS can interpolate between the specified values to determine the pressure profile. Simple linear interpolation is used to calculate the pressures on stages which are not explicitly specified.

You can enter the condenser and reboiler pressure drops explicitly within the appropriate operation property view. Default pressure drops for the condenser and reboiler are zero, and a non-zero value is not necessary to produce a converged solution.

If the pressure of a Column product stream (including side vapour or liquid draws, side stripper bottom streams, or internal stream assignments) is set (either by specification or calculation) prior to running the Column, HYSYS “backs” this value into the column and uses this value for the convergence process. If you do specify a stream pressure that allows HYSYS to calculate the column pressure profile, it is not necessary to specify another value within the column property view. If you later change the pressure of an attached stream, the Column is rerun.

Recall that whenever a change is made in a stream, HYSYS checks all operations attached to that stream and recalculates as required.

Number of Stages

The number of stages that you specify for the tray section does not include the condenser and bottom reboiler, if present. If sidestrippers are to be added to the column, their stages are not included in this number. By default, HYSYS numbers stages from the top down. If you want, you can change the numbering scheme to bottom-up by selecting this scheme on the Connections page of the Design tab.

HYSYS initially treats the stages as being ideal. If you want your stages to be treated as real stages, you must specify efficiencies on the Efficiencies page of the Parameters tab. Once you provide efficiencies for the stages, even if the value you specify is 1, HYSYS treats the stages as being real.

Stream

The feed stream and product stream location, conditions, and composition are treated as Basic Column Parameters during convergence.

2.2.4 Pressure Flow

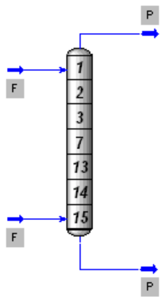
In the following sections, the pressure flow specifications presented are the recommended configurations if no other equipment, such as side strippers, side draws, heat exchanger, and so forth, are connected. Other combinations of pressure flow specifications are possible, however they can lead to less stable configurations.

Regardless of the pressure flow specification configuration, when performing detailed dynamic modeling it is recommended that at least valves be added to all boundary streams. Once valves have been added, the resulting boundary streams can all be specified with pressure specifications, and, where necessary, flow controlled with flow controllers.

Absorber

The basic Absorber column has two inlet and two exit streams. When used alone, the Absorber has four boundary streams and so requires four Pressure Flow specifications. A pressure specification is always required for the liquid product stream leaving the bottom of the column. A second pressure specification should be added to the vapour product of the column, with the two feed streams having flow specifications.

Figure 2.5



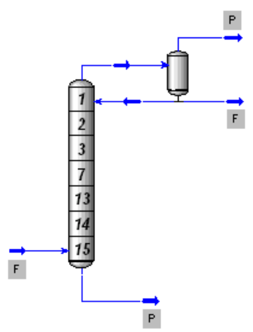
The column shows the recommended pressure flow specifications for a stand alone absorber column.

If there are down stream unit operations attached to the liquid product stream, then a column sump needs to be simulated. There are several methods for simulating the column sump. A simple solution is to use a reboiled absorber, with the reboiler duty stream specified as zero in place of the absorber. Another option is to feed the liquid product stream directly into a separator, and return the separator vapour product to the bottom stage of the column.

Refluxed Absorber

The basic Refluxed Absorber column has a single inlet and two or three exit streams, depending on the condenser configuration. When used alone, the Refluxed Absorber has three or four boundary streams (depending on the condenser) and requires four or five pressure-flow specifications; generally two pressure and three flow specifications. A pressure specification is always required for the liquid product stream leaving the bottom of the column. The extra specification is required due to the reflux stream and is discussed in **Section 2.7 - Column-Specific Operations**.

Figure 2.6



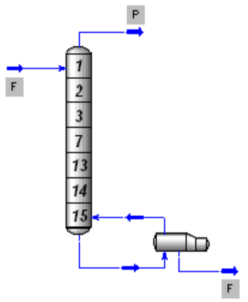
The column shows the recommended pressure flow specifications for a stand alone refluxed absorber with a partial condenser.

If there are down stream unit operations attached to the liquid product stream, then a column sump needs to be simulated. There are several methods for simulating the column sump. A simple solution is to use a distillation column, with the reboiler duty stream specified as zero in place of the refluxed absorber. Another option is to feed the liquid product stream directly into a separator, and return the separator vapour product to the bottom stage of the column.

Reboiled Absorber

A Reboiled Absorber column has a single inlet and two exit streams. When used alone, the Reboiled Absorber has three boundary streams and so requires three Pressure Flow specifications; one pressure and two flow specifications. A pressure specification is always required for the vapour product leaving the column.

Figure 2.7

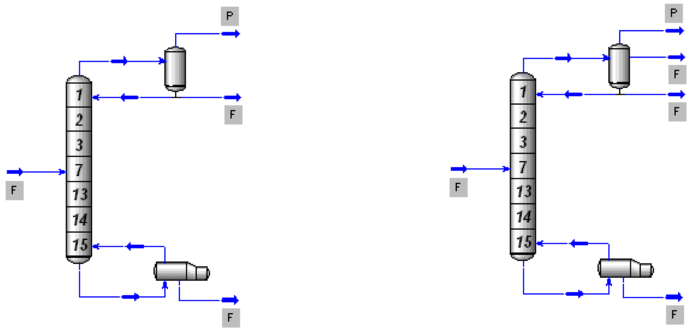


The column shows the recommended pressure flow specifications for a stand alone reboiled absorber.

Distillation Column

The basic Distillation column has one inlet and two or three exit streams, depending on the condenser configuration.When used alone, the Distillation column has three or four boundary streams but requires four or five pressure-flow specifications; generally one pressure and three or four flow specifications. The extra pressure-flow specification is required due to the reflux stream, and is discussed in **Section 2.7 - Column-Specific Operations**.

Figure 2.8



The column shows the recommended pressure-flow specifications for a stand alone distillation column with a partial condenser.

The column shows the recommended pressure-flow specifications for a stand alone three phase distillation column with a partial condenser.

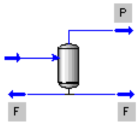
The Three Phase Distillation column is similar to the basic Distillation column except it has three or four exit streams. So when used alone, the Three Phase Distillation column has four to five boundary streams, but requires five or six pressure-flow specifications; generally one pressure and four to five flow specifications.

Condenser and Reboiler

The following sections provide some recommended pressure-flow specifications for simple dynamic modeling only. The use of flow specifications on reflux streams is not recommended for detailed modeling. If the condenser liquid level goes to zero, a mass flow specification results in a large volumetric flow because the stream is a vapour.

It is highly recommended that the proper equipment be added to the reflux stream (for example pumps, valves, and so forth). In all cases, level control for the condenser should be used to ensure a proper liquid level.

Partial Condenser

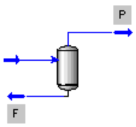


The partial condenser has three exit streams:

- overhead vapour
- reflux
- distillate

All three exit streams must be specified when attached to the main tray section. One pressure specification is recommended for the vapour stream, and one flow specification for either of the liquid product streams. The final pressure flow specification can be a second flow specification on the remaining liquid product stream, or the Reflux Flow/Total Liquid Flow value on the Specs page of the Dynamics tab of the condenser can be specified.

Fully-Refluxed Condenser

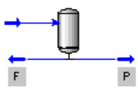


The Fully-Refluxed condenser has two exit streams:

- overhead vapour
- reflux

A pressure specification is required for the overhead vapour stream, and a flow specification is required for the reflux stream.

Total Condenser

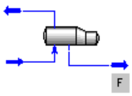


A Total condenser has two exit streams:

- reflux
- distillate

There are several possible configurations of pressure flow specifications for this type of condenser. A flow specification can be used for the reflux stream and a pressure flow spec can be used for the distillate stream. Two flow specifications can be used, however, it is suggested that a vessel pressure controller be setup with the condenser duty as the operating variable.

Reboiler



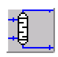
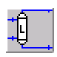
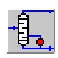
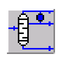
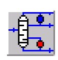
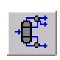
The Reboiler has two exit streams:

- boilup vapour
- bottoms liquid

Only one exit stream can be specified. If a pressure constraint is specified elsewhere in the column, this exit stream must be specified with a flow rate.

2.3 Column Installation

The first step in installing a Column is deciding which type you want to install. Your choice depends on the type of equipment (for example, reboilers and condensers) your Column requires. HYSYS has several basic Column templates (pre-constructed column configurations) which can be used for installing a new Column. The most basic Column types are described in the table below.

Basic Column Types	Icon	Description
Absorber		Tray section only.
Liquid-Liquid Extractor		Tray section only.
Reboiled Absorber		Tray section and a bottom stage reboiler.
Refluxed Absorber		Tray section and an overhead condenser.
Distillation		Tray section with both a reboiler and condenser.
Three Phase Distillation		Tray section, three-phase condenser, reboiler. Condenser can be either chemical or hydrocarbon specific.

There are two ways that you can add a basic Column type to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access UnitOps property view by pressing **F12**.
2. Click the **Prebuilt Columns** radio button.
3. From the list of available unit operations, select the column type.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.
2. Double-click the column type icon you want to install.

Refer to **Section 2.3.1 - Input Experts** for more information.

The Input Expert property view appears.

There are also more complex Column types, which are described in the table below.

Complex Column Types	Description
3 Sidestripper Crude Column	Tray section, reboiler, condenser, 3 sidestrippers, and 3 corresponding pump around circuits.
4 Sidestripper Crude Column	Tray section, reboiler, condenser, an uppermost reboiled sidestripper, 3 steam-stripped lower sidestrippers, and 3 corresponding pump around circuits.
FCCU Main Fractionator	Tray section, condenser, an upper pump around reflux circuit and product draw, a mid-column two-product-stream sidestripper, a lower pump around reflux circuit and product draw, and a quench pump around circuit at the bottom of the column.
Vacuum Reside Tower	Tray section, 2 side product draws with pump around reflux circuits and a wash oil-cooled steam stripping section below the flash zone.

To add a complex column type to your simulation:

1. In the **Flowsheet** menu, click the Add Operation command. The UnitOps property view appears.

   You can also access UnitOps property view by pressing **F12**.

2. Click the **Prebuilt Columns** radio button.
3. From the list of available unit operations, select the column type.
4. Click the **Add** button. The column property view appears.

2.3.1 Input Experts

Input Experts guide you through the installation of a Column. The Input Experts are available for the following six standard column templates:

- Absorber
- Liquid-Liquid Extractor
- Reboiled Absorber
- Refluxed Absorber
- Distillation
- Three Phase Distillation

Details related to each column template are outlined in **Section 2.3.2 - Templates**. Each Input Expert contains a series of input pages whereby you must specify the required information for the page before advancing to the next one. When you have worked through all the pages, you have specified the basic information required to build your column. You are then placed in the Column property view which gives comprehensive access to most of the column features.

Refer to **Chapter 12 - Session Preferences** in the **HYSYS User Guide** for details on how to access the Session Preferences property view.

It is not necessary to use the Input Experts to install a column. You can disable and enable the Input Experts option on the **Options** page in the **Simulation** tab of the Session Preferences property view.

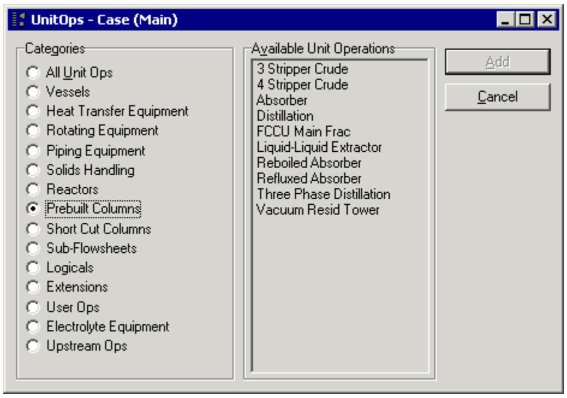
If you do not use the Input Experts, you move directly to the Column property view when you install a new column.

2.3.2 Templates

HYSYS contains a number of column templates which have been designed to simplify the installation of columns.

A Column Template is a pre-constructed configuration or “blueprint” of a common type of Column, including Absorbers, Reboiled and Refluxed Absorbers, Distillation Towers, and Crude Columns. A Column Template contains the unit operations and streams that are necessary for defining the particular column type, as well as a default set of specifications.

Figure 2.9



All Column templates can be viewed by opening the UnitOps property view and selecting the Prebuilt Columns radio button.

When you add a new Column, HYSYS gives you a choice of the available templates. Simply select the template that most closely matches your column configuration, provide the necessary input in the Input Expert property view (if applicable), and HYSYS installs the equipment and streams for you in a new Column subflowsheet. Stream connections are already in place, and HYSYS provides default names for all internal streams and equipment. You can then make modifications by adding, removing or changing the names of any streams or operations to suit your specific requirements.

Clicking the Side Ops button on the final page of the Column Input Expert opens the Side Operations Input Expert wizard, which guides you through the process of adding a side operation to your column.

In addition to the basic Column Templates which are included with HYSYS, you can create custom Templates containing Column configurations that you commonly use.

HYSYS Column Conventions

Column Tray Sections, Overhead Condensers, and Bottom Reboilers are each defined as individual unit operations. Condensers and Reboilers are not numbered stages, as they are considered to be separate from the Tray Section.

By making the individual components of the column separate pieces of equipment, there is easier access to equipment information, as well as the streams connecting them.

The following are some of the conventions, definitions, and descriptions of the basic columns:

Column Component	Description
Tray Section	A HYSYS unit operation that represents the series of equilibrium trays in a Column.
Stages	Stages are numbered from the top down or from the bottom up, depending on your preference. The top tray is 1, and the bottom tray is N for the top-down numbering scheme. The stage numbering preference can be selected on the Connections page of the Design tab on the Column property view.
Overhead Vapor Product	The overhead vapour product is the vapour leaving the top tray of the Tray Section in simple Absorbers and Reboiled Absorbers. In Refluxed Absorbers and Distillation Towers, the overhead vapour product is the vapour leaving the Condenser.
Overhead Liquid Product	The overhead liquid product is the Distillate leaving the Condenser in Refluxed Absorbers and Distillation Towers. There is no top liquid product in simple Absorbers and Reboiled Absorbers.

Column Component	Description
Bottom Liquid Product	The bottom liquid product is the liquid leaving the bottom tray of the Tray Section in simple Absorbers and Refluxed Absorbers. In Reboiled Absorbers and Distillation Columns, the bottom liquid product is the liquid leaving the Reboiler.
Overhead Condenser	An Overhead Condenser represents a combined Cooler and separation stage, and is not given a stage number.
Bottom Reboiler	A Bottom Reboiler represents a combined heater and separation stage, and is not given a stage number.

Default Replaceable Specifications

Replaceable specifications are the values, which the Column convergence algorithm is trying to meet. When you select a particular Column template, or as you add side equipment, HYSYS creates default specifications. You can use the specifications that HYSYS provides, or replace these specifications with others more suited to your requirements.

The available default replaceable specifications are dependent on the Basic Column type (template) that you have chosen. The default specifications for the four basic column templates are combinations of the following:

- Overhead vapour flowrate
- Distillate flowrate
- Bottoms flowrate
- Reflux ratio
- Reflux rate

Refer to the **Monitor Page** and **Specs Page** in **Section 2.4.1 - Design Tab** for more information.

The specifications in HYSYS can be set as specifications or changed to estimates.

The provided templates contain only pre-named internal streams (streams which are both a feed and product). For instance, the Reflux stream, which is named by HYSYS, is a product from the Condenser and a feed to the top tray of the

Tray Section.

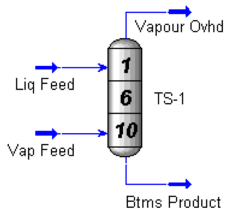
The pressure for a tray section stage, condenser or reboiler can be specified at any time on the Pressures page of the Column property view.

In the following schematics, you specify the feed and product streams, including duty streams.

Absorber Template

The only unit operation contained in the Absorber is the Tray Section, and the only streams are the overhead vapour and bottom liquid products.

Figure 2.10



A schematic representation of the Absorber.

There are no available specifications for the Absorber, which is the base case for all tower configurations. The conditions and composition of the column feed stream, as well as the operating pressure, define the resulting converged solution. The converged solution includes the conditions and composition of the vapour and liquid product streams.

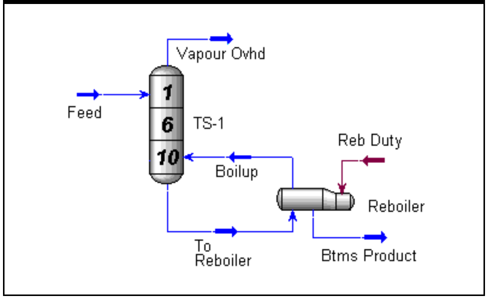
The Liquid-Liquid Extraction Template is identical to the Absorber Template.

The remaining Column templates have additional equipment, thus increasing the number of required specifications.

Reboiled Absorber Template

The Reboiled Absorber template consists of a tray section and a bottom reboiler. Two additional streams connecting the Reboiler to the Tray Section are also included in the template.

Figure 2.11

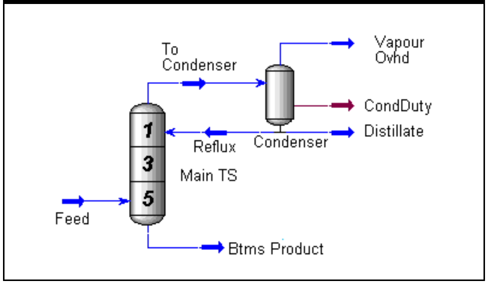


When you install a Reboiled Absorber (in other words, add only a Reboiler to the Tray Section), you increase the number of required specifications by one over the Base Case. As there is no overhead liquid, the default specification in this case is the overhead vapour flow rate.

Refluxed Absorber Template

The Refluxed Absorber template contains a Tray Section and an overhead Condenser (partial or total). Additional material streams associated with the Condenser are also included in the template. For example, the vapour entering the Condenser from the top tray is named to Condenser by default, and the liquid returning to the Tray Section is the Reflux.

Figure 2.12



When you install a Refluxed Absorber, you are adding only a Condenser to the base case. Specifying a partial condenser increases the number of required specifications by two over the Base Case. The default specifications are the overhead vapour flow rate, and the side liquid (Distillate) draw. Specifying a total condenser results in only one available specification, since there is no overhead vapour product.

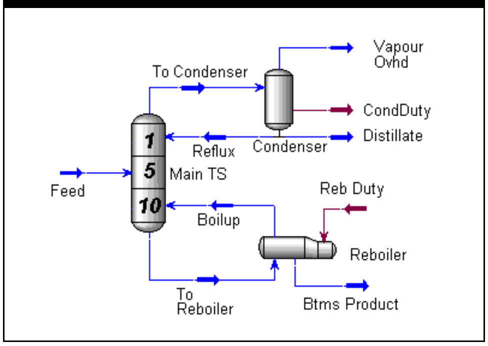
Either of the overhead vapour or distillate flow rates can be specified as zero, which creates three possible combinations for these two specifications. Each combination defines a different set of operating conditions. The three possible Refluxed Absorber configurations are listed below:

- Partial condenser with vapour overhead but no side liquid (distillate) draw.
- Partial condenser with both vapour overhead and distillate draws.
- Total condenser with distillate but no vapour overhead draw.

Distillation Template

If you select the Distillation template, HYSYS creates a Column with both a Reboiler and Condenser. The equipment and streams in the Distillation template are therefore a combination of the Reboiled Absorber and Refluxed Absorber Templates

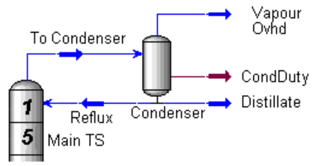
Figure 2.13



Reflux Ratio

The number of specifications for a column with both a Reboiler and Condenser depends on the condenser type. For a partial condenser, you must specify three specifications. For a total condenser, you must specify two specifications. The third default specification (in addition to Overhead Vapor Flow Rate and Side Liquid Draw) is the Reflux Ratio.

Figure 2.14



$$\text{Reflux Ratio} = \frac{\text{Reflux}}{\text{Vapour Ovhd} + \text{Distillate}}$$

The Reflux Ratio is defined as the ratio of the liquid returning to the tray section divided by the total flow of the products (see the figure above). If a water draw is present, its flow is not included in the ratio.

As with the Refluxed Absorber, the Distillation template can have either a Partial or Total Condenser. Choosing a Partial Condenser results in three replaceable specifications, while a Total Condenser results in two replaceable specifications.

The pressure in the tower is, in essence, a replaceable specification, in that you can change the pressure for any stage from the Column property view.

The pressure remains fixed during the Column calculations.

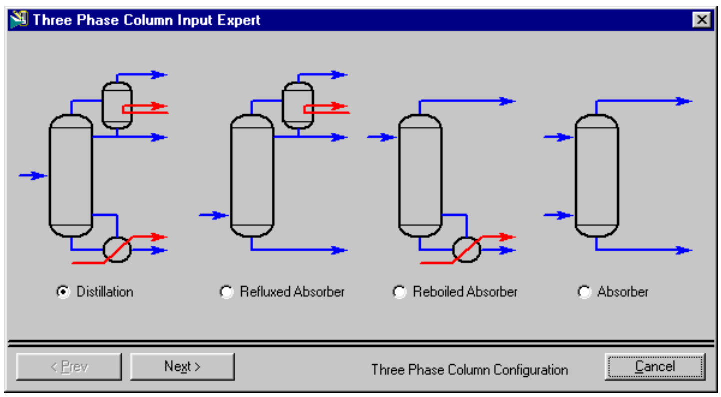
The following table gives a summary of replaceable column (default) specifications for the basic column templates.

Templates	Vapour Draw	Distillate Draw	Reflux Ratio
Reboiled Absorber	X		
Refluxed Absorber			
Total Condenser		X	
Partial Condenser	X		X
Distillation			
Total Condenser		X	X
Partial Condenser	X	X	X

Three Phase Distillation Template

If you select the Three Phase Distillation template, HYSYS creates a Column based on a three phase column model.

Figure 2.15



The same standard column types exist for a three phase system that are available for the "normal" two phase (binary) systems.

Using the Three Phase Column Input Expert, the initial property view allows you to select from the following options:

- Distillation
- Refluxed Absorber
- Reboiled Absorber
- Absorber

Each choice builds the appropriate column based on their respective standard (two phase) system templates.

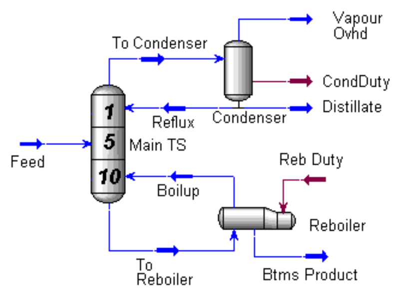
If the Input Expert is turned off, installing a Three Phase column template opens a default Column property view for a Distillation type column equipped with a Reboiler and Condenser.

The key difference between using the standard column templates and their three phase counterparts lies in the solver that is used. The default solver for three phase columns is the “Sparse Continuation” solver which is an advanced solver designed to handle three phase, non ideal chemical systems, that other solvers cannot.

When using the Three Phase Column Input Expert some additional specifications can be required when compared with the standard (binary system) column setups.

Figure 2.16

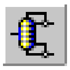
It requires some expertise to set up, initialize, and solve three phase distillation problems. Additional modeling software applications such as DISTIL, use residue curve maps and distillation region diagrams to determine feasible designs, and can greatly assist in the initial design work. Contact your local AspenTech representative for details.



Clicking the Side Ops button on the final page of the Three Phase Column Input Expert opens the Side Operations Input Expert wizard, which guides you through the process of adding a side operation to your column.

2.4 Column Property View

The column property view is sectioned into tabs containing pages with information pertaining to the column. The column property view is accessible from the main flowsheet or Column subflowsheet.



Column Runner icon

In the Column subflowsheet, the column property view is also known as the Column Runner, and can be accessed by clicking the Column Runner icon.

The column property view is used to define specifications, provide estimates, monitor convergence, view stage-by-stage and product stream summaries, add pump-arounds and side-strippers, specify dynamic parameters and define other Column parameters such as convergence tolerances, and attach reactions to column stages.

The column property view is essentially the same when accessed from the main flowsheet or Column subflowsheet. However, there are some differences:

- The Connections page in the main flowsheet column property view displays and allows you to change all product and feed stream connections. In addition, you can specify the number of stages and condenser type.
- The Connections page in the subflowsheet Column property view (Column Runner) allows you to change the product and feed stream connections, and gives more flexibility in defining new streams.
- In the main flowsheet Column property view, the Flowsheet Variables and Flowsheet Setup pages allow you to specify the transfer basis for stream connections, and permit you to view selected column variables.

In order to make changes or additions to the Column in the main simulation environment, the Solver should be active. Otherwise HYSYS cannot register your changes.

Column Convergence

Refer to the section on the **Specification Tolerances for Solver** for more information.

The Run and Reset buttons are used to start the convergence algorithm and reset the Column, respectively. HYSYS first performs iterations toward convergence of the inner and outer loops (Equilibrium and Heat/Spec Errors), and then checks the individual specification tolerances.

The Monitor page displays a summary of the convergence procedure for the Equilibrium and Heat/Spec Errors. An example of a converged solution is shown in the following figure:

Figure 2.17

Iter	Step	Equilibrium	Heat / Spec
9	1.0000	0.000010	0.002915
10	1.0000	0.000004	0.000950
11	1.0000	0.000003	0.001709
12	1.0000	0.000005	0.001983
13	1.0000	0.000002	0.000369

A summary of each of the tabs in the Column property view are in the following sections.

2.4.1 Design Tab

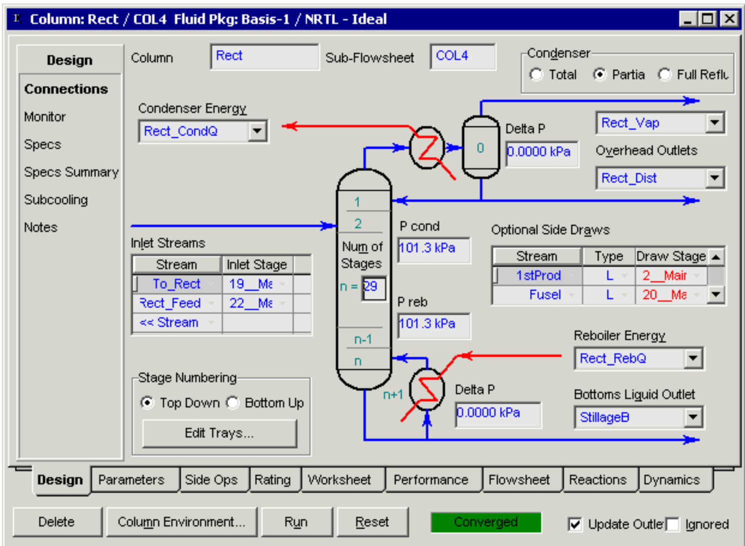
The following sections detail information regarding the Column property view pages. All pages are common to both the Main Column property view and the Column Runner, unless stated otherwise.

Column Runner is another name for the subflowsheet Column property view.

Connections page (Main Flowsheet)

The main flowsheet Connections page allows you to specify the name and location of feed streams, the number of stages in the tray section, the stage numbering scheme, condenser type, names of the Column product streams, and Condenser/Reboiler energy streams.

Figure 2.18



If you have modified the Column Template (for example if you added an additional Tray Section), the Connections page appears differently than what is shown in Figure 2.18.

The streams shown in this property view reside in the parent or main flowsheet; they do not include Column subflowsheet streams, such as the Reflux or Boilup. In other words, only feed and product streams (material and energy) appear on this page.

When the column has complex connections, the Connections page changes to the property view shown in the figure below.

Figure 2.19

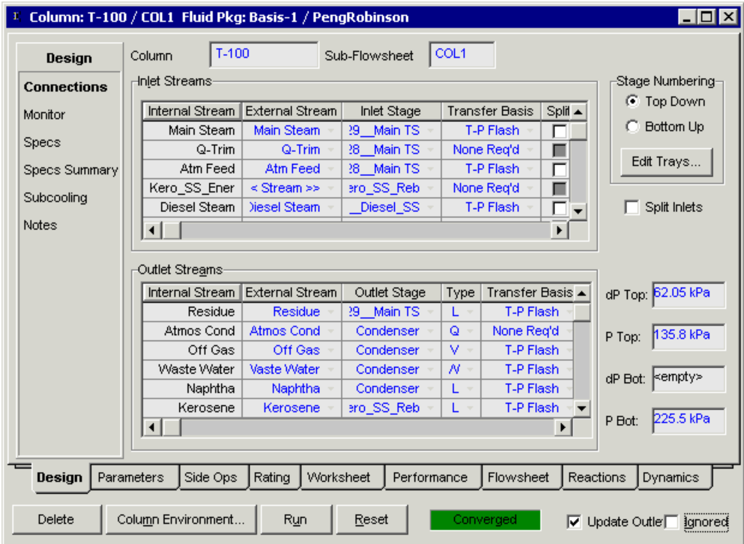
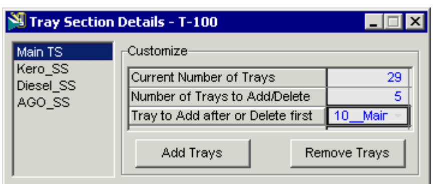


Figure 2.19 is an example of the Connections page for a Stripper Crude.

You can also split the feed streams by selecting the **Split** checkbox associated to the stream.

Click the **Edit Trays** button to open the Tray Section Details property view. You can edit the number of trays in the column, and add or delete trays after or before the tray number of your choice in this property view.

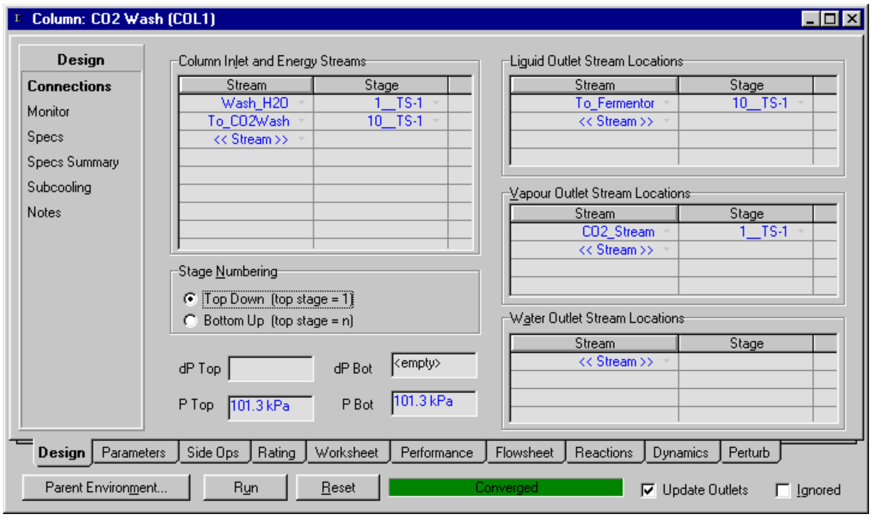
Figure 2.20



Connections page (Column Runner)

The Connections page displayed in the Column Runner (inside the Column subflowsheet) appears as shown in the following figure.

Figure 2.21



If you specify a new stream name in any of the cells, this creates the stream inside the Column. This new stream is not automatically transferred into the main flowsheet.

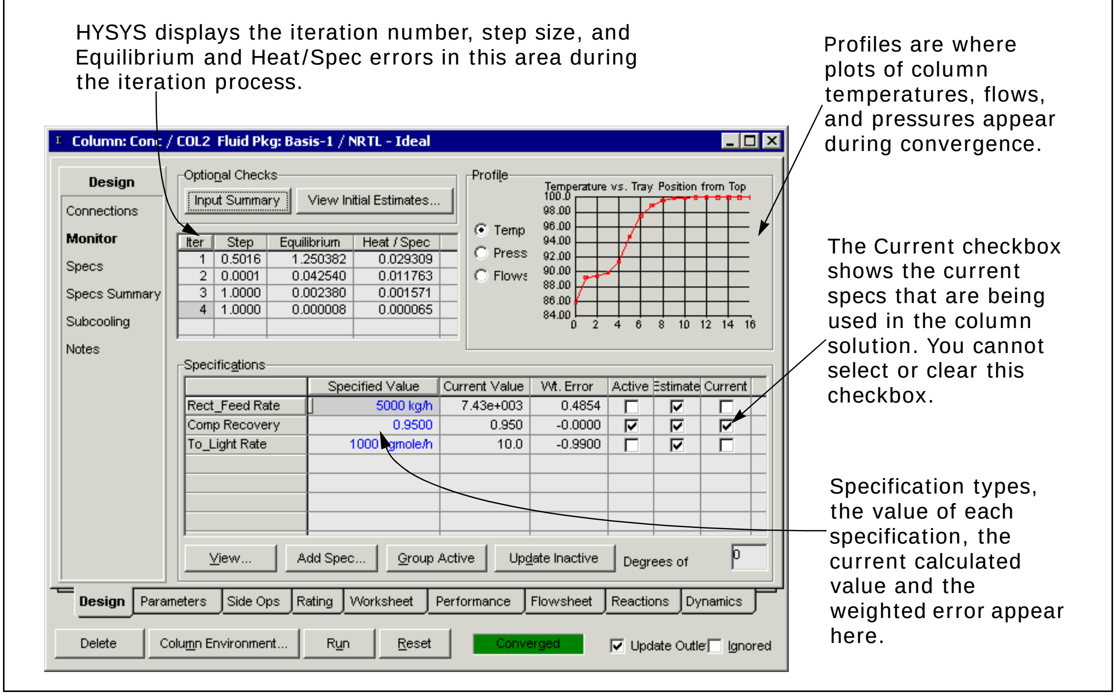
All feed and energy streams, as well as the associated stage, appear in the left portion of the Connections page. Liquid, vapour, and water product streams and locations appear on the right side of the page.

You can connect or disconnect streams from the Connections page, as well as change the stream location.

Monitor Page

The Monitor page is primarily used for editing specifications, monitoring Column convergence, and viewing Column profile plots. An input summary, and a property view of the initial estimates can also be accessed from this page.

Figure 2.22



Optional Checks Group

In the Optional Checks group, you find the following two buttons:

Refer to **Section 1.3 - Object Status & Trace Windows** in the **HYSYS User Guide** for details concerning the Trace Window.

Button	Function
Input Summary	Provides a column input summary in the Trace Window. The summary lists vital tower information including the number of trays, the attached fluid package, attached streams, and specifications. You can click the Input Summary button after you make a change to any of the column parameters to view an updated input summary. The newly defined column configuration appears.
View Initial Estimates	Opens the Summary page of the Column property view, and displays the initial temperature and flow estimates for the column. You can then use the values generated by HYSYS to enter estimates on the Estimates page. These estimates are generated by performing one iteration using the current column configuration. If a specification for flow or temperature has been provided, it is honoured in the displayed estimates.

Profile Group

During the column calculations, a profile of temperature, pressure or flow appears, and is updated as the solution progresses. Select the appropriate radio button to display the desired variable versus tray number profile.

Specifications Group

Each specification, along with its specified value, current value, weighted error, and status is shown in the Specifications group.

Refer to **Section 2.5 - Column Specification Types** for a description of the available specification types.

You can change a specified value by typing directly in the associated Specified Value cell. Specified values can also be viewed and changed on the Specs and Specs Summary pages. Any changes made in one location are reflected across all locations.

New specifications can also be added via the Specs page.

Double-clicking on a cell within the row for any listed specification opens its property view. In this property view, you can define all the information associated with a particular specification. Each specification property view has three tabs:

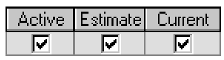
- Parameters
- Summary
- Spec Type

Further details are outlined in the section on the **Specification Property View**.

This property view can also be accessed from both the Specs and Specs Summary pages.

Spec Status Checkboxes

Figure 2.23



The status of listed specifications are one of the following types:

Status	Description
Active	The active specification is one that the convergence algorithm is trying to meet. An active specification always serves as an initial estimate (when the **Active** checkbox is selected, HYSYS automatically selects the **Estimate** and **Current** checkboxes). An active specification always exhausts one degree of freedom. An Active specification is one which the convergence algorithm is trying to meet initially. An Active specification has the **Estimate** checkbox selected also.
Estimate	An Estimate is considered an Inactive specification because the convergence algorithm is not trying to satisfy it. To use a specification as an estimate only, clear the **Active** checkbox. The value then serves only as an initial estimate for the convergence algorithm. An estimate does not exhaust an available degree of freedom. An Estimate is used as an initial “guess” for the convergence algorithm, and is considered to be an Inactive specification.

Status	Description
Current	This checkbox shows the current specs being used by the column solution. When the **Active** checkbox is selected, the **Current** checkbox is automatically selected. You cannot alter this checkbox. When Alternate specs are used and an existing hard to solve spec has been replaced with an Alternate spec, this checkbox shows you the current specs used to solve the column. A Current specification is one which is currently being used in the column solution.
Completely Inactive	To disregard the value of a specification entirely during convergence, clear both the **Active** and **Estimate** checkboxes. By ignoring a specification rather than deleting it, you are always able to use it later if required. The current value appears for each specification, regardless of its status. An Inactive specification is therefore ideal when you want to monitor a key variable without including it as an estimate or specification. A Completely Inactive specification is ignored completely by the convergence algorithm, but can be made Active or an Estimate at a later time.

The degrees of freedom value appears in the Degrees of Freedom field on the Monitor page. When you make a specification active, the degrees of freedom is decreased by one. Conversely, when you deactivate a specification, the degrees of freedom is increased by one. You can start column calculations when there are zero degrees of freedom.

Variables such as the duty of the reboiler stream, which is specified in the Workbook, or feed streams that are not completely known can offset the current degrees of freedom. If you feel that the number of active specifications is appropriate for the current configuration, yet the degrees of freedom is not zero, check the conditions of the attached streams (material and energy).You must provide as many specifications as there are available degrees of freedom. For a simple Absorber there are no available degrees of freedom, therefore no specifications are required. Distillation columns with a partial condenser have three available degrees of freedom.

Specification Group Buttons

The four buttons which align the bottom of the Specifications group allow you to manipulate the list of specs. The table below describes the four buttons.

Refer to the section on the **Specification Property View** for more details.

Refer to **Section 2.5 - Column Specification Types** for a description of the available specification types.

Button	Action
View	Move to one of the specification cells and click the **View** button to display its property view. You can then make any necessary changes to the specification. To change the value of a specification only, move to the Specified Value cell for the specification you want to change, and type in the new value. You can also double-click in a specification cell to open its property view.
Add Spec	Opens the Column Specifications menu list, from which you can select one or multiple (by holding the **CTRL** key while selecting) specifications, and then click the Add Spec(s) button. The property view for each new spec is shown and its name is added to the list of existing specifications.
Update Inactive	Updates the specified value of each inactive specification with its current value.
Group Active	Arranges all active specifications together at the top of the specifications list.

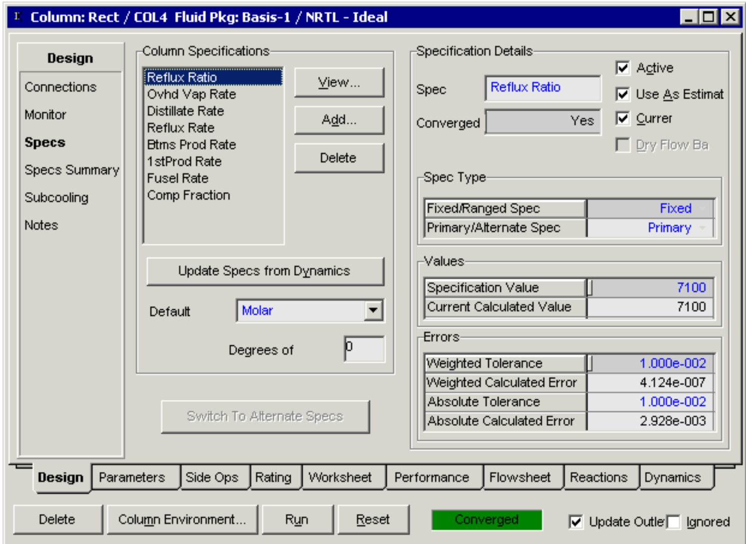
Specs Page

Refer to the **Default Replaceable Specifications** in **Section 2.3.2 - Templates** for more information.

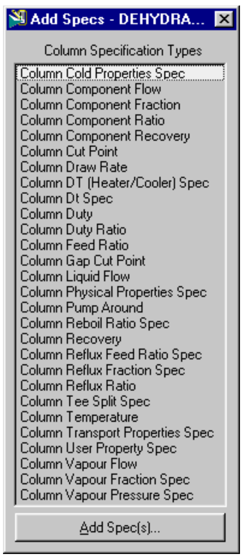
Adding and changing Column specifications is straightforward. If you have created a Column based on one of the templates, HYSYS already has default specifications in place. The type of default specification depends on which of the templates you have chosen.

The active specification values are used as initial estimates when the column initially starts to solve.

Figure 2.24



Column Specifications Group



Add Specs property view

The following buttons are available:

Button	Action
View	Opens the property view for the highlighted specification. Alternatively, you can object inspect a spec name and select **View** from the menu. Refer to the section on the **Specification Property View** for more details.
Add	Opens the Add Specs property view, from which you can select one or multiple (by holding the **CTRL** key while selecting) specifications, and then click the **Add Spec(s)** button. The property view for each new spec is shown, and its name is added to the list of existing specifications. Refer to **Section 2.5 - Column Specification Types** for a description of the available specification types.
Delete	Removes the highlighted specification from the list.

From the Default Basis drop-down list, you can choose the basis for the new specifications to be Molar, Mass or Volume.

The Update Specs from Dynamics button replaces the specified value of each specification with the current value (lined out value) obtained from Dynamic mode.

Specification Property View

Figure 2.25 is a typical property view of a specification. In this property view, you can define all the information associated with a particular specification. Each specification property view has three tabs:

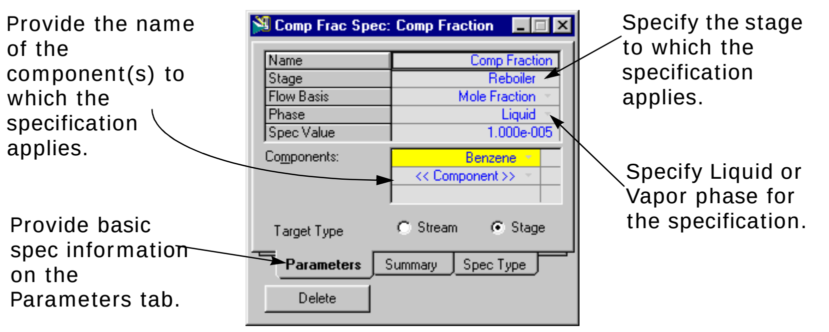
- Parameters
- Summary
- Spec Type

This example shows a component recovery specification which requires the stage number, spec value, and phase type when a Target Type of *Stage* is chosen.

Specification information is shared between this property view, and the specification list on both the Monitor and Specs Summary pages. Altering information in one location automatically updates across all other locations.

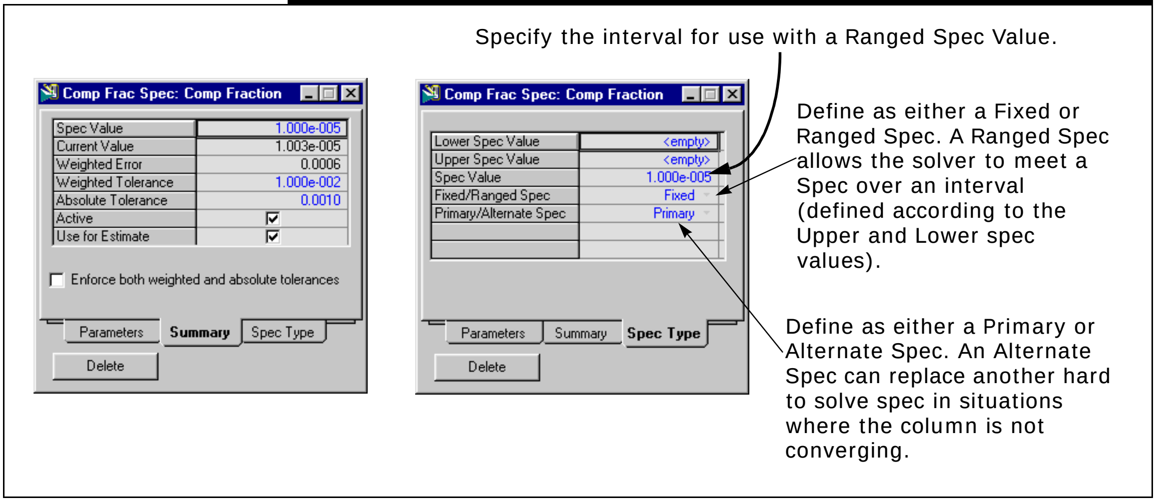
For example, you can enter the spec value in one location, and the change is reflected across all other locations.

Figure 2.25



The Summary tab is used to specify tolerances, and define whether the specification is Active or simply an Estimate.

Figure 2.26



The Spec Type tab (as shown in **Figure 2.26**) can be used to define specifications as either Fixed/Ranged and Primary/ Alternate. By default, all specifications are initially defined as Fixed and Primary. Advanced solving options available in HYSYS allow the use of both Alternate and Ranged Spec types.

The following section further details the advanced solving options available in HYSYS.

Ranged and Alternate Specs

The reliability of any solution method depends on its ability to solve a wide group of problems. Some specs like purity, recovery, and cut point are hard to solve compared to a flow or reflux ratio spec. The use of Alternate and/or Ranged Specs can help to solve columns that fail due to difficult specifications.

> **If the Column solves on an Alternate or Ranged Spec, the status bar reads "Converged - Alternate Specs" highlighted in purple.**

Refer to **Advanced Solving Options Button** in **Section 2.4.2 - Parameters Tab** for further details.

Configuration of these advanced solving options are made by selecting the Advanced Solving Options button located on the Solver page. The advanced solving options are only available for use with either the Hysim I/O or Modified I/O solving methods.

Fixed/Ranged Specs

For a Fixed Spec, HYSYS attempts to solve for a specific value. For a Ranged Spec, the solver attempts to meet the specified value, but if the rest of the specifications are not solved after a set number of iterations, the spec is perturbed within the interval range provided for the spec until the column converges.

When the solver attempts to meet a Ranged spec, the Wt. Error becomes zero when the Current Value is within the Ranged interval (as shown on the Monitor page).

Any column specification can be specified over an interval. A Ranged Spec requires both lower and upper specification values to be entered. This option (when enabled), can help solve columns where some specifications can be varied over an interval to meet the rest of the specifications.

Primary/Alternate Specs

A Primary Spec must be met for the column solution to converge. An Alternate Spec can be used to replace an existing hard to solve specification during a column solution. The solver first attempts to meet an active Alternate spec value, but if the rest of the specifications are not solved after a minimum number of iterations, the active Alternate spec is replaced by an inactive Alternate spec.

When an existing spec is replaced by an alternate spec during a column solution, the Current checkbox is cleared for the original (not met) spec and is selected for the alternate spec.

The number of active Alternate specs must always equal the number of inactive Alternate specs.

This option (when enabled), can help solve columns where some specifications can be ignored (enabling another) to meet the rest of the specifications and converge the column.

Both Ranged or Alternate Specs must be enabled and configured using the Advanced Solving Options Button located on the Solver page of the Parameters tab before they can be applied during a column solution.

Specification Tolerances for Solver

For more information, refer to **Section 2.4.2 - Parameters Tab**.

The Solver Tolerances feature allows you to specify individual tolerances for your Column specifications. In addition to HYSYS converging to a solution for the Heat/Spec and Equilibrium Errors, the individual specification tolerances must also be satisfied. HYSYS first performs iterations until the Heat/Spec (inner loop), and Equilibrium (outer loop) errors are within specified tolerances.

The Column specifications do not have individual tolerances during this initial iteration process; the specification errors are “lumped” into the Heat/Spec Error. Once the Heat/Spec and Equilibrium conditions are met, HYSYS proceeds to compare the error with the tolerance for each individual specification. If any of these tolerances are not met, HYSYS iterates through the Heat/Spec, and Equilibrium loops again to produce another converged solution. The specification errors and tolerances are again compared, and the process continues until both the inner/outer loops and the specification criteria are met.

Specific Solver Tolerances can be provided for each individual specification. HYSYS calculates two kinds of errors for each specification:

- an absolute error
- a weighted error

When the Weighted and Absolute Errors are less than their respective tolerances, an Active specification has converged.

The absolute error is simply the absolute value of the difference between the calculated and specified values:

$$Error_{absolute} = |Calculated\ Value - Specified\ Value| \quad (2.3)$$

The Weighted Error is a function of a particular specification type. When a specification is active, the convergence algorithm is trying to meet the Weighted Tolerances (Absolute Tolerances are only used if no Weighted Tolerances are specified, or the weighted tolerances are not met).

Therefore, both the weighted and absolute errors must be less than their respective tolerances for an active specification to converge. HYSYS provides default values for all specification tolerances, but any tolerance can be changed. For example, if you are dealing with ppm levels of crucial components, composition tolerances can be set tighter (smaller) than the other specification tolerances. If you delete any tolerances, HYSYS cannot apply the individual specification criteria to that specification, and **Ignore** appears in the tolerance input field.

The specification tolerance feature is simply an “extra” to permit you to work with individual specifications and change their tolerances if desired.

Specification Details Group

For a highlighted specification in the Column Specifications group, the following information appears:

- Spec Name
- **Convergence Condition**. If the weighted and absolute errors are within their tolerances, the specification has converged and Yes appears.
- **Status**. You can manipulate the Active and Use As Estimate checkboxes.
- **Dry Flow Basis**. Draw specifications are calculated on a dry flow basis by selecting the **Dry Flow Basis** checkbox.

  This option is only available for draw specifications. The checkbox is greyed out if it does not apply to the specification chosen.

Refer to the **Spec Status Checkboxes** for further details concerning the use of these checkboxes.

Refer to the section on the **Ranged and Alternate Specs** for more details.

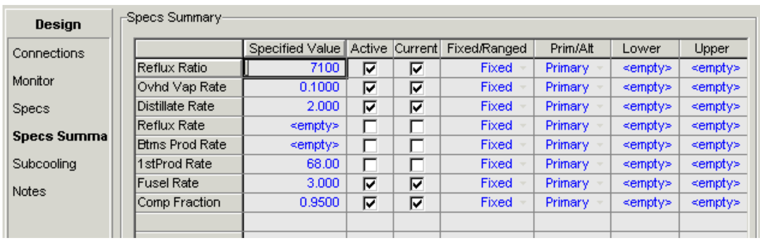
- **Spec Type**. You can select between Fixed/Ranged and Primary/Alternate specs.
- Specified and Current Calculated Values.
- Weighted/Absolute Tolerance and Calculated Error.

You can edit any specification values (in the Column property view) shown in blue.

Specs Summary Page

The Specs Summary page lists all Column specifications available along with relevant information. This specification information is shared with the Monitor page and Specs page. Altering information in one location automatically updates across all other locations.

Figure 2.27



Refer to the section on the **Specification Property View** for more details.

You can edit any specification details shown in blue.

You can double-click in a specification cell to open its property view.

Subcooling Page

The Subcooling page allows you to specify subcooling for products coming off the condenser of your column. You can specify the condenser product temperature or the degrees to subcool. For columns without condensers, such as absorbers, this page requires no additional information.

The Subcooling page is not available for Liquid-Liquid Extractor.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page/Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

2.4.2 Parameters Tab

The Parameters tab shows the column calculation results, and is used to define some basic parameters for the Column solution. The Parameters tab consists of six pages:

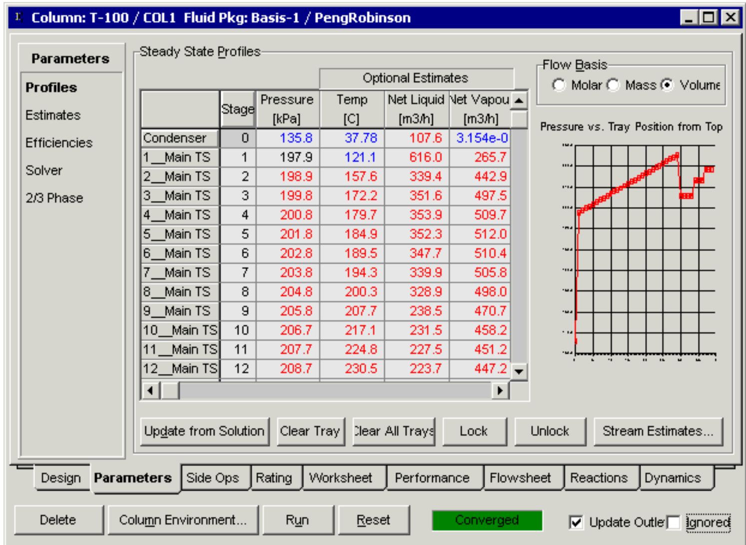
- Profiles
- Estimates
- Efficiencies
- Solver
- 2/3 Phase
- Amines

Profiles Page

The Profiles page shows the column pressure profile, and provides estimates for the temperature, net liquid and net vapour flow for each stage of the column. You can specify tray estimates in the Temperature column, Net Liquid column and Net Vapour column, or view the values calculated by HYSYS.

The graph in **Figure 2.28** depicts the pressure profile across the column.

Figure 2.28



Use the radio buttons in the Flow Basis group to select the flow type you want displayed in the Net Liquid and Net Vapour columns. The Flow Basis group contains three radio buttons:

- Molar
- Mass
- Volume

At least one iteration must have occurred for HYSYS to convert between bases. In this way, values for the compositions on each tray are available.

The buttons in the Steady State Profiles group are defined as follows:

Button	Function
Update from Solution	Transfers the current values that HYSYS has calculated for the trays into the appropriate cells. Estimates that have been Locked (displayed in blue) are not updated. The Column Profiles page on the Performance tab allows you to view all the current values.
Clear	Deletes values for the selected tray.

Button	Function
Clear All Trays	Deletes values for all trays.
Lock	Changes all red values (unlocked estimates, current values, interpolated values) to blue (locked), which means that they cannot be overwritten by current values when the Update from Solution button is clicked.
Unlock	Changes all blue values (locked) to red (unlocked). Unlocked values are overwritten by current values when the Update from Solution button is clicked.
Stream Estimates	Displays the temperature, molar flow, and enthalpy of all streams attached to the column operation.

Although the Profiles page is mainly used for steady state simulation, it does contain vital information for running a column in dynamics. One of the most important aspects of running a column in dynamics is the pressure profile. While a steady state column can run with zero pressure drop across a tray section, the dynamic column requires a pressure drop. In dynamics, an initial pressure profile is required before the column can run. This profile can be from the steady state model or can be added in dynamics. If a new tray section is created in Dynamic mode, the pressure profile can be obtained from the streams if not directly specified. In either case, the closer the initial pressure profile is to the one calculated while running in dynamics, the fewer problems you encounter.

Estimates Page

The Estimates page allows you to view and specify composition estimates.

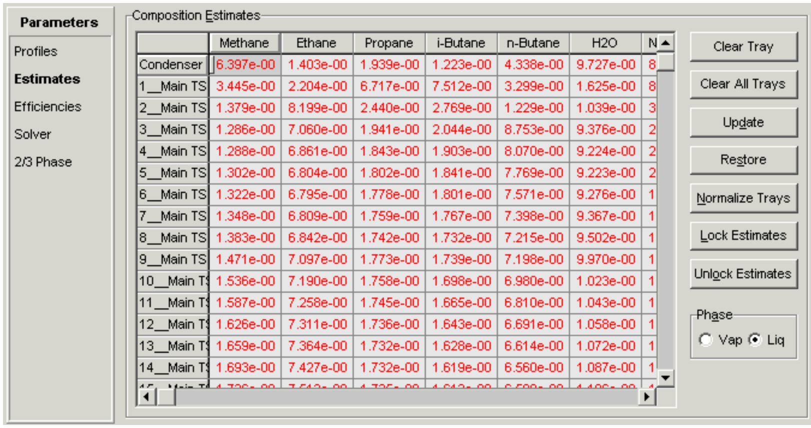
To see the initial estimates generated by HYSYS, click the View Initial Estimates button on the Monitor page.

When you specify estimates on stages that are not adjacent to each other, HYSYS cannot interpolate values for intermediate stages until the solution algorithm begins.

Estimates are NOT required for column convergence.

You can specify tray by tray component composition estimates for the vapour phase or liquid phase. Each composition estimate is on a mole fraction basis, so values must be between 0 and 1.

Figure 2.29



HYSYS interpolates intermediate tray component values when you specify compositions for non-adjacent trays. The interpolation is on a log basis. Unlike the temperature estimates, the interpolation for the compositions does not wait for the algorithm to begin. Select either the Vap or Liq radio button in the Phase group to display the table for the vapour or liquid phase, respectively.

The Composition Estimates group has the following buttons:

Button	Action
Clear Tray	Deletes all values, including user specified (blue) and HYSYS generated (red), for the selected tray. HYSYS does not ask for confirmation before deleting estimates.
Clear All Trays	Deletes all values for all trays. HYSYS does not ask for confirmation before deleting estimates.
Update	Transfers the current values which HYSYS has calculated for tray compositions into the appropriate cells. Estimates that have been locked (shown in blue) cannot be updated.

Button	Action
Restore	Removes all HYSYS updated values from the table, and replaces them with your estimates and their corresponding interpolated values. Any cells that did not contain estimates or interpolated values are shown as <empty>. This button essentially reverses the effect of the Update button. If you had entered some estimate values, click the Unlock Estimates button, and click the Update button. All the values in the table appear in red. You can restore your estimated values by clicking the Restore button.
Normalize Trays	Normalizes the values on a tray so that the total of the composition fractions equals 1. HYSYS ignores <empty> cells, and normalizes the compositions on a tray provided that there is at least one cell containing a value.
Lock Estimates	Changes all red values (unlocked estimates, current values, interpolated values) to blue (locked), which means that they cannot be overwritten by current values when the Update button is clicked.
Unlock Estimates	Changes all blue values (locked) to red (unlocked). Unlocked values are overwritten by current values when the Update button is clicked.

Efficiencies Page

The Efficiencies page allows you to specify Column stage efficiencies on an overall or component-specific basis. Efficiencies for a single stage or a section of stages can easily be specified.

Refer to the section on **Special Case - Amines Property Package** for more information.

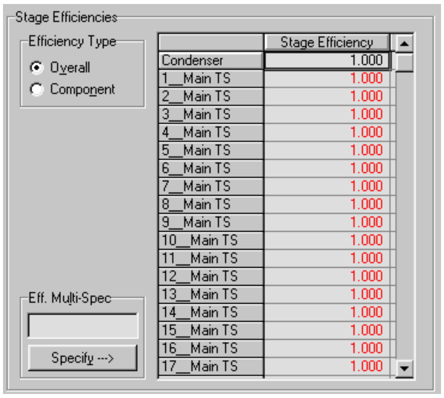
Fractional efficiencies cannot be given for the condenser or reboiler stages, nor should they be set for feed or draw stages.

The functionality of this page is slightly different when working with the Amines Property Package.

HYSYS uses a modified Murphree stage efficiency. All values are initially set to 1.0, which is consistent with the assumption of ideal equilibrium or theoretical stages. If this assumption is not valid for your column, you have the option of specifying the number of actual stages, and changing the efficiencies for one or more stages.

To specify an efficiency to multiple cells, highlight the desired cells, enter a value in the Eff. Multi-Spec field, and click the Specify button.

Figure 2.30



The data table on the Efficiency page gives a stage-by-stage efficiency summary.

The efficiencies are fractional. In other words, an efficiency of 1.0 corresponds to 100% efficiency.

Overall stage efficiencies can be specified by selecting the Overall radio button in the Efficiency Type group, and entering values in the appropriate cells.

Component-specific efficiencies can be specified by selecting the Component radio button, and entering values in the appropriate cells.

Special Case - Amines Property Package

For more information on the Amines Property Packages, refer to **Appendix C - Amines Property Package** of the **HYSYS Simulation Basis** guide.

When solving a column for a case using the Amines Property Package, HYSYS always uses stage efficiencies for H2S and CO2 component calculations. If these are not specified on the Efficiencies page of the Column property view, HYSYS calculates values based on the tray dimensions. Tray dimensions can be specified on the Amines page of the Parameters tab. If column dimensions are not specified, HYSYS uses its default tray values to determine the efficiency values.

If you specify values for the CO2 and H2S efficiencies, these are the values that HYSYS uses to solve the column. If you want to solve the column again using efficiencies generated by HYSYS, click the **Reset H2S, CO2** button, which is available on the **Efficiencies** page. Run the column again, and HYSYS calculates and displays the new values for the efficiencies.

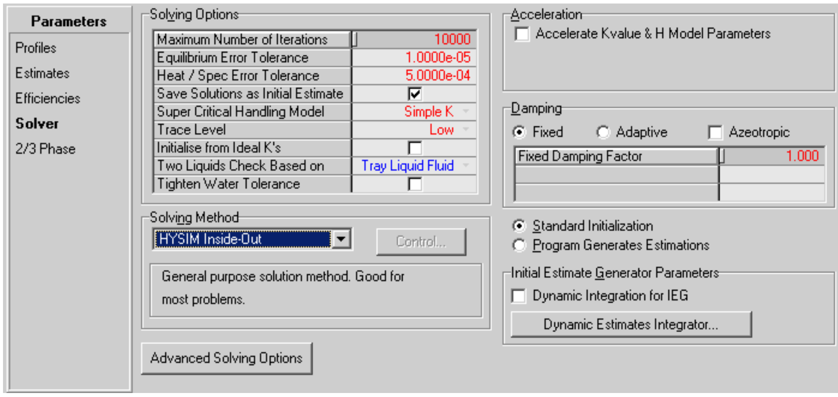
Select the **Transpose** checkbox to change the component efficiency matrix so that the rows list components and the columns list the stages.

The Reset H2S, CO2 button, and the Transpose checkbox are available only if the Efficiency Type is set to Component.

Solver Page

You can manipulate how the column solves the column variables on the Solver page.

Figure 2.31

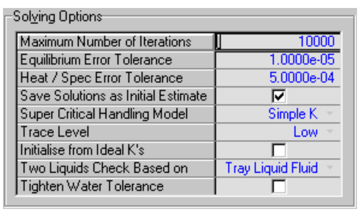


The Solving Method Group, Acceleration Group, and Damping Group will have different information displayed according to the options selected within the group.

Solving Options Group

Specify your preferences for the column solving behaviour in the Solving Options group.

Figure 2.32



Maximum Number of Iterations

The Column convergence process terminates if the maximum number of iterations is reached. The default value is 10000, and applies to the outer iterations. If you are using Newton's method, and the inner loop does not converge within 50 iterations, the convergence process terminates.

Equilibrium and Heat/Spec Tolerances

Convergence tolerances are pre-set to very tight values, thus ensuring that regardless of the starting estimates (if provided) for column temperatures, flow rates, and compositions, HYSYS always converges to the same solution. However, you have the option of changing these two values if you want. Default values are:

- **Inner Loop**. Heat and Spec Error: 5.000e-04
- **Outer Loop**. Equilibrium Error: 1.000e-05

Because the default values are already very small, you should use caution in making them any smaller. You should not make these tolerances looser (larger) for preliminary work to reduce computer time. The time savings are usually minor, if any. Also, if the column is in a recycle or adjust loop, this could cause difficulty for the loop convergence.

Equilibrium Error

The value of the equilibrium error printed during the column iterations represents the error in the calculated vapour phase mole fractions. The error over each stage is calculated as *one minus the sum of the component vapour phase mole fractions*. This value is then squared; the total equilibrium error is the sum of the squared values. The total equilibrium error must be less than 0.00001 to be considered a converged column.

Heat and Spec Error

The heat and specification error is the sum of the absolute values of the heat error and the specification error, summed over each stage in the tower.

This total value is divided by the number of inner loop equations. The heat error contribution is the heat flow imbalance on each tray divided by the total average heat flow through the stage.

The specification error contribution is the sum of each individual specification error divided by an appropriate normalization factor.

- For component(s) flow, the normalization factor is the actual component(s) flow.
- For composition, it is the actual mole fraction.
- For vapour pressure and temperature, it is a value of 5000.
- And so forth.

The total sum of heat and spec errors must be less than 0.0005 to be considered a converged column.

The allowed equilibrium error and heat and spec error are tighter than in most programs, but this is necessary to avoid meta-stable solutions, and to ensure satisfactory column heat and material balances.

Save Solution as Initial Estimate

This option is on by default, and it saves converged solutions as estimates for the next solution.

Super Critical Handling Model

Supercritical phase behaviour occurs when one or more Column stages are operating above the critical point of one or more components. During the convergence process, supercritical behaviour can be encountered on one or more stages in the Column. If HYSYS encounters supercritical phase behaviour, appropriate messages appear in the Trace Window.

HYSYS cannot use the equation of state or activity model in the supercritical range, so an alternate method must be used. You can specify which method you want HYSYS to use to model the phase behaviour. There are three choices for supercritical calculations:

Refer to **Section 1.3 - Object Status & Trace Windows** in the **HYSYS User Guide** for details on the Trace Window.

Model	Description
Simple K	The default method. HYSYS calculates K-values for the components based on the vapour pressure model being used. Using this method, the K-values which are calculated are ideal K-values.
Decrease Pressure	When supercritical conditions are encountered, HYSYS reduces the pressure on all trays by an internally determined factor, which can be seen in the Trace Window when the Verbose option is used. This factor is gradually decreased until supercritical conditions no longer exist on any tray, at which point, the pressure in the column is gradually increased to your specified pressure. If supercritical conditions are encountered during the pressure increase, the pressure is once again reduced and the process is repeated.
Adjacent Tray	When supercritical conditions are encountered on a tray, HYSYS searches for the closest tray above which does not have supercritical behaviour. The non-supercritical conditions are substituted in the phase calculations for the tray with supercritical conditions.

Trace Level

The Trace Level defines the level of detail for messages displayed in the Trace Window, and can be set to Low, Medium, or High. The default is Low.

Initialize from Ideal K's

When this checkbox is selected, HYSYS initializes its column solution using ideal K values which are calculated from vapour pressure correlations. The ideal K-value option, which is also used by HYSIM, increases the compatibility between HYSIM and HYSYS.

By default, the **Initialize from Ideal K's** checkbox is cleared. HYSYS uses specified composition estimates or generates estimates to rigorously calculate K-values.

Two Liquids Check Based on

This option allows you to specify a check for two liquid phases in the column. The check is based on one of the following criteria:

- **No 2 Liq Check**. Disables the two liquid check.
- **Tray Liquid Fluid**. The calculation is based on the composition of the liquid in the column.
- **Tray Total Fluid**. The calculation is based on the overall composition of the fluid in the column.

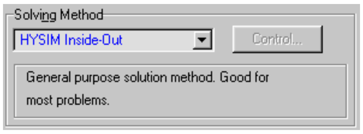
Tighten Water Tolerance

When this checkbox is selected, HYSYS increases the contribution of the water balance error to the overall balance error in order to solve columns with water more accurately. The default setting for this checkbox is cleared.

Solving Method Group

The Solving Method drop-down list allows you to select the column solution method.

Figure 2.33



The display field, which appears below the drop-down list, provides explanations for each method, and is restated here:

Method	Explanation
HYSIM Inside-Out	General purpose method, which is good for most problems.
Modified HYSIM Inside-Out	General purpose method, which allows mixer, tee, and heat exchangers inside the column subflowsheet. Only a simple Heat Exchanger Model (Calculated from Column) is available in the Column subflowsheet. The Simple Rating, End-Point, and Weighted models are not available.
Newton Raphson Inside-Out	General purpose method, which allows liquid-phase kinetic reactions inside the Column subflowsheet.
Sparse Continuation Solver	An equation based solver. It supports two liquid phases on the trays, and its main use is for solving highly non-ideal chemical systems and reactive distillation.
Simultaneous Correction	Simultaneous method using dogleg methods. Good for chemical systems. This method also supports reactive distillation.
OLI Solver	Only used to calculate the column unit operation in an electrolyte system.

Inside-Out

With the “inside-out” based algorithms, simple equilibrium and enthalpy models are used in the inner loop to solve the overall component and heat balances as well as any specifications. The outer loop updates the simple thermodynamic models with rigorous model calculations.

Open the Trace Window at the bottom of the HYSYS Desktop to view messages regarding the convergence of the column.

General Features of the Solving Methods

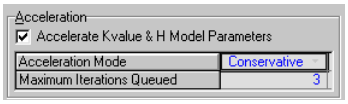
The following table displays the general features of all the HYSYS column solving methods.

	HYSIM I/O	Modified HYSIM I/O	Newton Raphson I/O	Sparse Continuation	Simultaneous Correction	OLI
Component Efficiency Handling	Yes	Yes	No	Yes	No	Yes
Total Efficiency Handling	Yes	Yes	No	Yes	No	Yes
Additional Side Draw	Yes	Yes	Yes	Yes	Yes	Yes
Vapour Bypass	Yes	Yes	No	Yes	No	No
Pump Arounds	Yes	Yes	No	Yes	No	Yes
Side Stripper	Yes	Yes	No	Yes	No	No
Side Rectifier	Yes	Yes	No	Yes	No	No
Mixer & Tee in Sub-flowsheet	No	Yes	No	No	No	No
Three Phase	Yes (water draw)	Yes (water draw)	No	Yes	No	Yes
Chemical (reactive)	No	No	Yes	Yes	Yes	Internal reactions

Acceleration Group

When selected, the **Accelerate K value & H Model Parameters** checkbox displays two fields, which relate to an acceleration program called the Dominant Eigenvalue Method (DEM).

Figure 2.34



By default, the Accelerate K value & H Model Parameters checkbox is cleared.

The DEM is a numerical solution program, which accelerates convergence of the simple model K values and enthalpy parameters. It is similar to the Wegstein accelerator, with the main difference being that the DEM considers all interactions between the variables being accelerated. The DEM is applied independently to each stage of the column.

Use the acceleration option if you find that the equilibrium error is decreasing slowly during convergence. This should help to speed up convergence. Notice that the Accelerate K value & H Model Parameters checkbox should NOT be selected for AZEOTROPIC columns, as convergence tends to be impeded.

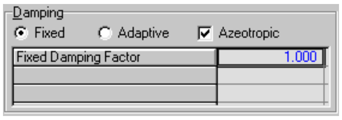
The listed DEM parameters include:

Parameter	Description
Acceleration Mode	Select either Conservative or Aggressive. With the Conservative approach, smaller steps are taken in the iterative procedure, thus decreasing the chance of a bad step.
Maximum Iterations Queued	Allows you to choose the number of data points from previous iterations that the accelerator program uses to obtain a solution.

Damping Group

Choose the Damping method by selecting either the Fixed or Adaptive radio button.

Figure 2.35



Fixed Damping

If you select the Fixed radio button, you can specify the damping factor. The damping factor controls the step size used in the outer loop when updating the simple thermodynamic models used in the inner loop. For the vast majority of hydrocarbon-oriented towers, the default value of 1.0 is appropriate, which permits a full adjustment step. However, should you encounter a tower where the heat and specification errors become quite small, but the equilibrium errors diverge or oscillate and converge very slowly, try reducing the damping factor to a value between 0.3 and 0.9. Alternatively, you could enable Adaptive Damping, allowing HYSYS to automatically adjust this factor.

Changing the damping factor has an effect on problems where the heat and spec error does not converge.

There are certain types of columns, which definitely require a special damping factor.

Use the following table as a guideline in setting up the initial value.

Type of Column	Damping Factor
All hydrocarbon columns from demethanizers to debutanizers to crude distillation units	1.0
Non-hydrocarbon columns including air separation, nitrogen rejection	1.0

Type of Column	Damping Factor
Most petrochemical columns including C2= and C3= splitters, BTX columns	1.0
Amines absorber	1.0
Amines regenerator, TEG strippers, sour water strippers	0.25 to 0.50
Highly non-ideal chemical columns without azeotropes	0.25 to 0.50
Highly non-ideal chemical columns with azeotropes	0.50 to -1.0*

The Azeotropic checkbox on the Solver page of the Parameters tab must be selected for an azeotropic column to converge.

As shown in the table above, an azeotropic column requires the azeotrope checkbox to be enabled. There are two ways to indicate to HYSYS that you are simulating an azeotropic column:

- Enter a negative damping factor, and HYSYS automatically selects the **Azeotropic** checkbox.

The absolute value of the damping factor is always displayed.

- Enter a positive value for the damping factor, and select the **Azeotropic** checkbox.

Adaptive Damping

If you select the Adaptive radio button, the Damping matrix displays three fields. HYSYS updates the damping factor as the column solution is calculated, depending on the Damping Period and convergence behaviour.

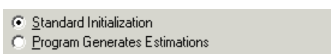
Damping Period	Description
Initial Damping Factor	Specifies the starting point for adaptive damping.
Adaptive Damping Period	The default Adaptive Damping Period is ten. In this case, after the tenth iteration, HYSYS looks at the last ten errors to see how many times the error increased rather than decreased. If the error increased more than the acceptable tolerance, this is an indication that the convergence is likely cycling, and the current damping factor is then multiplied by 0.7. Every ten iterations, the same analysis is done to see if the damping factor should be further decreased. Alternatively, if the error increased only once in the last period, the damping factor is increased to allow for quicker convergence.
Reset Initial Damping Factor	If this checkbox is selected, the current damping factor is used the next time the column is solved. If this checkbox is cleared, the damping factor before adaptive damping was applied is used.

Initialization Algorithm Radio Buttons

There are two types of method for the initialization algorithm calculation:

- **Standard Initialization** radio button uses the tradition initialization algorithm in Hysys.
- **Program Generates Estimations** radio button uses a new functionality that handles the cases where the traditional initialization does not.

Figure 2.36



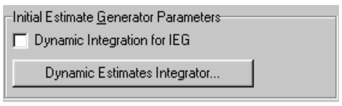
The following list situations when the Program Generates Estimations (PGE) initialization method is used:

- The PGE initialization handles systems with more than 25 components while the standard initialization does not handle systems with more than 25 components without the user's initial estimation.
- When column does not converge with the standard initialization method (default), switching to PEG may converge the column. The new algorithm eliminates the discrepancy in the temperature and component estimates, which may exist in standard initialization.

Initial Estimate Generator Parameters

You can enable the initial estimate generator (IEG) by selecting the **Dynamic Integration for IEG** checkbox. The IEG then performs iterative flash calculations (NRSolver, PV, and PH) to provide initial estimates for the temperature and composition profiles. No user estimates are required when the **Dynamic Integration for IEG** checkbox is selected.

Figure 2.37

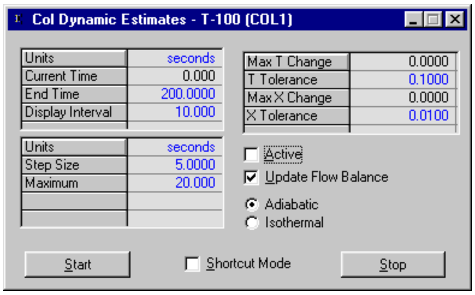


Click the Dynamic Estimates Integrator button, and the Col Dynamic Estimates property view appears as shown in the figure below.

Col Dynamic Estimates Property View

The Col Dynamic Estimates property view allows you to further define the dynamic estimates parameters.

Figure 2.38



You can set parameters for the time period over which the dynamic estimates are calculated, as well as set the calculation tolerance. A selected **Active** checkbox indicates that the Dynamic Integration for IEG is on. Select either the Adiabatic or Isothermal radio button to set the dynamic initialization flash type.

If you want to generate the dynamic estimates without running the column, you can do so from this property view by clicking the Start button. If you want to stop calculations before the specified time has elapsed, click the Stop button. You do not have to manually click the Start button to generate the estimates; if the Dynamic Integration for IEG option is active, HYSYS generates them automatically whenever the column is running.

The Shortcut Mode checkbox allows you to bypass this step once a set of estimates is generated, that is, once the column has converged.

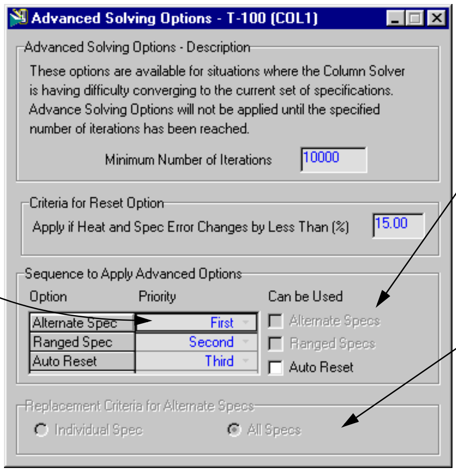
If you are running simulation with an iterative solving procedure where the column has to be calculated several times, it is a good idea to select this option to save on calculation time.

Advanced Solving Options Button

When you click the Advanced Solving Options button, the Advanced Solving Options property view appears.

Figure 2.39

The order in which the solving options are executed is based on the priority.



The "use" checkboxes must be selected in order to enable a particular option.

These checkboxes are only enabled if the corresponding spec type exists.

All the Alternate active specs can be replaced on an individual spec basis or all specs simultaneously. The alternative (active) spec with the larger error is replaced with an alternative inactive spec with minimum error.

If the Column converges on an Alternate or Ranged Spec, the status bar reads "Converged - Alternate Specs" highlighted in purple.

On the Advanced Solving Options property view, each solving option (for exmple, Alternate, Ranged, and Autoreset) has a solving priority and also a checkbox option. To use a particular solving option, you have to select the corresponding checkbox. You must also specify the priority of the solving method. This is the order in which the solving options are executed (either first, second or third).

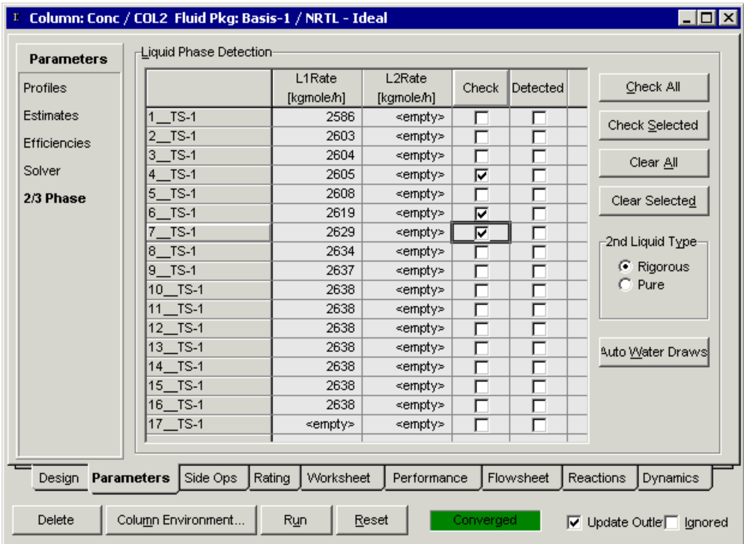
Advanced solving options cannot be used until the minimum number of iterations are met. If the column is not solved after the minimum number of iterations, the solver switches to an advanced solving option according to the solving priority. This process is repeated until all the solving options have been attempted or the column converges.

When a column is in recycle, by default, the solver switches to the original set of specs after each recycle iteration or the next time the column solves.

2/3 Phase Page

The 2/3 Phase page is relevant only when you are working with three-phase distillation. On this page, you can check for the presence of two liquid phases on each stage of your column.

Figure 2.40



This page is not available for Liquid-Liquid Extractor.

The Liquid Phase Detection table lists the liquid molar flow rates on each tray of the tray section, including the reboiler and the condenser.

In order for HYSYS to check for two liquid phases on any given stage, select the checkbox in the **Check** column. If a second liquid phase is calculated, this is indicated in the Detected column, and by a calculated flowrate value in the L2Rate column. The buttons in the Liquid Phase Detection group serve as aids in selecting and de-selecting the trays you want to check.

Checking for liquid phases in a three phase distillation tower greatly increases the solution time. Typically, checking the top few stages only, provides reasonable results.

The 2nd Liquid Type group allows you to specify the type of calculation HYSYS performs when checking for a second liquid phase. When the Pure radio button is selected, HYSYS checks only for pure water as the second phase. This helps save calculation time when working with complex hydrocarbon systems. When you want a more rigorous calculation, select the Rigorous radio button.

By default, HYSYS selects Pure for all hydrocarbon, and Rigorous for all chemical based distillations. This default selection criteria is based on the type of fluid package used but you can always change it.

Auto Water Draws Button

The Auto Water Draws (AWD) option allows for the automatic adding and removing of total aqueous phase draws depending on the conditions in the converged column.

The Auto Water Draws facility is available for IO and MIO solvers.

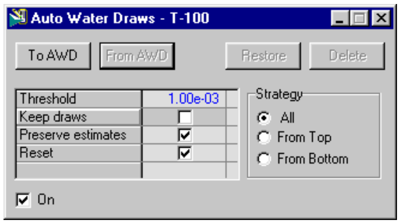
AWD updating process is based on direct check of stage fluid phases. The direct check follows the Two Liquids Check Based on control criteria for detecting the aqueous phase. AWD mode is not available if No 2 Liq Check option is selected.

The Two Liquids Check Based on drop-down list is located in the Parameters tab of the Solver page in the Solving Options group.

To manipulate the AWD option, click the Auto Water Draws button to open the Auto Water Draws property view.

The Auto Water Draws button is available in both column subflowsheet and main flowsheet.

Figure 2.41



The Auto Water Draws property view contains the following objects:

Object	Description
On	Select this checkbox to activate the Auto Water Draws mode.
Threshold	The threshold value allows variation of the condition for 2nd liquid phase. The default value in this cell is 0.001 (same as for **Two Liquids Check based on** control). If you delete the value in this cell, the threshold is set to minimum possible value.
Keep draws	If this checkbox is selected, the added draws are not removed.
Preserve estimates	If this checkbox is selected, the converged values are preserved as estimates for the next column run.
Reset	If this checkbox is selected, the column Reset option is performed before each column run.
Strategy	There are three options of strategy to select from in the Strategy group: • **All**. All required changes in water draw configuration are done simultaneously. • **From Top**.Updates on the topmost stage from required is performed. • **From Bottom**. Updates on the bottommost stage from required is performed. The **All** option results typically in multiple water draws with small flows, and the From Top or From Bottom option results typically in fewer water draws.
To AWD	All existing water draws are converted to AWDs.

Object	Description
From AWD	Converts all AWDs to regular draws.
Restore	Restores the last successful (in other words, column equation were solved) AWD configuration.
Delete	Deletes all AWDs.

Two more columns are added in the table on the 2/3 Phase page when in Auto Water Draws mode. These two columns are called AWD and No AWD

- Set AWD mode for attached water draw by selecting the checkboxes under the **AWD** column.
- If the checkbox in the **No AWD** column is selected, no AWD will be attached to corresponding stage.

Amines Page

For more information on the Amines Property Package, refer to the **Appendix C - Amines Property Package** in the **HYSYS Simulation Basis** guide.

The Amines page appears on the Parameters tab only when working with the Amines Property Package.

The Amines Property Package is an optional property package that must be purchased in addition to the base version of HYSYS.

There are two groups on the Amine page:

- Tray Section Dimensions for Amine Package
- Approach to Equilibrium Results

Tray Section Dimensions for Amine Package

When solving the column using the Amines package, HYSYS always takes into account the tray efficiencies, which can either be user-specified, on the Efficiencies page, or calculated by HYSYS. Calculated efficiency values are based on the tray dimensions specified. The Amines page lists the tray section dimensions of your column, where you can specify these values that are used to determine the tray efficiencies in the Tray Section Dimensions for Amine Package group.

The list includes:

- Tray Section
- Weir Height
- Weir Length
- Tray Volume
- Tray Diameter

If tray dimensions are not specified, HYSYS uses the default tray dimensions to determine the efficiency values.

Approach to Equilibrium Results

Approach to Equilibrium values are used for the design, operation, troubleshooting, and de-bottlenecking for the absorption and regeneration columns in an amine plant. When you are modeling an amine column in HYSYS, you can calculate the Approach to Equilibrium values after the column converges. With this capability, you can adjust the flowrate of amine to achieve a certain Approach to Equilibrium value recommended by literature or in-house experts for the amine column. The extension is compatible with all of the major amine and mixtures of amines (in other words, MEA, DEA, TEA, DGA, DIPA, MDEA, and any mixtures of these amines).

The extension can only be used on a pre-converged amine treating unit simulation with the Amine Property Package.

The Approach to Equilibrium extension calculates the Approach to Equilibrium value of rich amine from the bottom of the absorber column in two methods:

- Partial Pressure
- Amine Molar Loading

Method 1 Partial Pressure

In this method, the Approach to Equilibrium is defined as the partial pressure of the acid gas in the rich amine stream exiting the absorber relative to the partial pressure of the acid gas in the main feed gas stream entering the absorber.

The Approach to Equilibrium calculation are as follows:

$$\text{H2S} = 100\% \cdot \left(\frac{\text{ppH2S}_{\text{rich amine exiting the absorber}}}{\text{ppH2S}_{\text{feed gas entering the absorber}}}\right) \tag{2.4}$$

$$\text{CO2} = 100\% \cdot \left(\frac{\text{ppCO2}_{\text{rich amine exiting the absorber}}}{\text{ppCO2}_{\text{feed gas entering the absorber}}}\right) \tag{2.5}$$

The Approach to Equilibrium results based on H2S and CO2 are expressed in percentages. When both H2S and CO2 are present, the highest Approach to Equilibrium percentage is usually reported, although both values should be reported.

Amine Molar Loading

Amine Molar Loading is defined as the loading of the rich amine solution leaving the absorber divided by the equilibrium amine loading, assuming that the amine is at equilibrium with the feed gas and is at the same temperature as the rich amine leaving the absorber. The temperature of the rich amine and the amine in equilibrium with the feed gas are the same. The result is expressed as a percentage as follows:

$$\begin{matrix}\text{Approach}\\ \text{to Equilibrium}\end{matrix} = 100\% \cdot \left(\frac{\begin{matrix}\text{Rich amine loading}\\ \text{in mole AG}\end{matrix} \Big/ \text{mole of amine}}{\begin{matrix}\text{Equilibrium loading}\\ \text{in mole AG}\end{matrix} \Big/ \text{mole of amine}}\right) \tag{2.6}$$

In general, the Approach to Equilibrium value calculated by the Partial Pressure method is greater than the one calculated by the Amine Molar Loading method.

2.4.3 Side Ops Tab

Side strippers, side rectifiers, pump arounds, and vapour bypasses can be added to the Column from this tab. To install any of these Side Operations, click the Side Ops Input Expert button or on the appropriate Side Ops page, click the Add button.

- If you are using the Side Ops Input Expert, a wizard guides you through the entire procedure of adding a side operation to your column.
- If you are using the Add button, complete the form which appears, and then click the Install button.

  Specifications that are created when you add a side operation are automatically added to the Monitor page and Specs page.

  For instance, when you add a side stripper, product draw and boilup ratio specs are added. As well, all appropriate operations are added; for example, with the side stripper (reboiled configuration), a side stripper tray section and reboiler are installed in the Column subflowsheet.

You can view or delete any Side Operation simply by positioning the cursor in the same line as the Operation, and clicking the **View** or **Delete** button.

If you are specifying Side Operations while in the Main simulation environment, make sure that the Solver is Active. Otherwise, HYSYS cannot register your changes.

The Side Ops tab is not available in the Liquid-Liquid Extractor.

Some solver methods do not allow side ops.

Refer to the table in the section on the **General Features of the Solving Methods** for more information.

Side Strippers Page

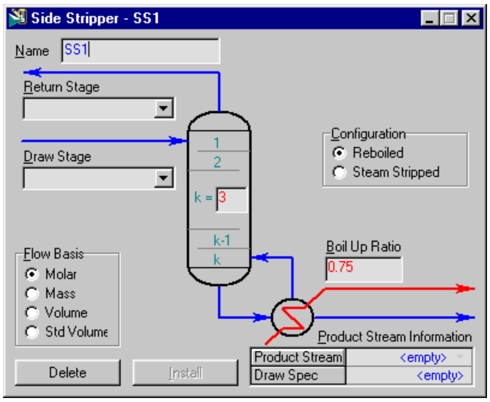
You can install a reboiled or steam-stripped side stripper on this page. You must specify the number of stages, the liquid draw stage (from the Main Column), the vapour return stage (to the Main Column), and the product stream and flow rate (on a molar, mass or volume basis).

For the reboiled configuration, you must specify the boilup ratio, which is the ratio of the vapour to the liquid leaving the reboiler.

For the steam-stripped configuration it is necessary to specify the steam feed.

The property view of the side stripper is shown in the figure below.

Figure 2.42



When you click the Install button, a side stripper tray section is installed, as well as a reboiler if you selected the Reboiled configuration.

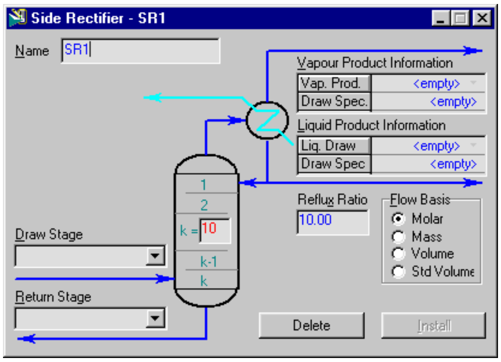
By default, the tray section is named SS1, the reboiler is named SS1_Reb, and the reboiler duty stream is named SS1_Energy. As you add additional Side strippers, the index increases (for example SS2, SS3, and so forth).

To change the side stripper draw and return stages from the Column property view, the Solver must be Active in the Main simulation environment.

Side Rectifiers Page

As with the side stripper, you must specify the number of stages, the liquid draw stage, and the vapour return stage.

Figure 2.43



The vapour and liquid product rates, as well as the reflux ratio are also required. These specifications are added to the Monitor page and Specs page of the Column property view.

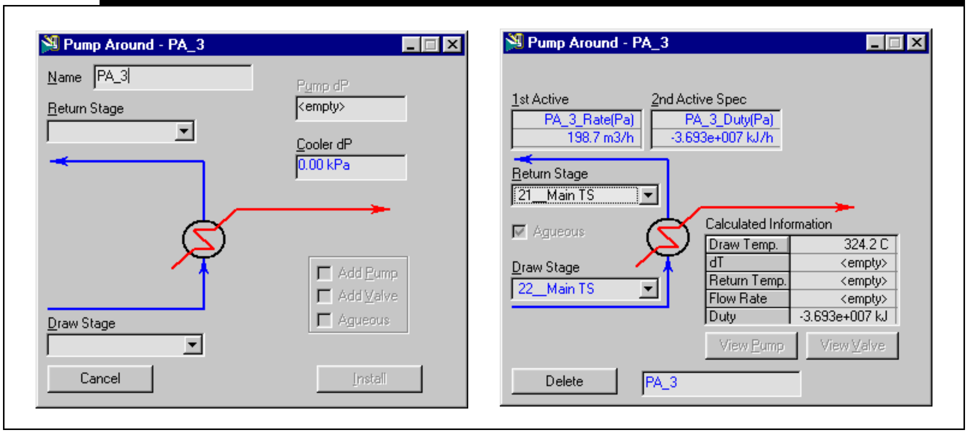
When you install the side rectifier, a side rectifier tray section and partial condenser are added. By default, the tray section is named SR_1, the condenser is named SR_1_Cond, and the condenser duty stream is named SR_1_Energy.

Pump Arounds Page

When you install the pump around, a Cooler is also installed. The default pump around specifications are the pump around rate and temperature drop. These are added on the Monitor page and Specs page of the Column property view.

After you click the Install button, the Pump Around property view changes significantly, as shown in the figure below, allowing you to change pump around specifications, and view pump around calculated information.

Figure 2.44



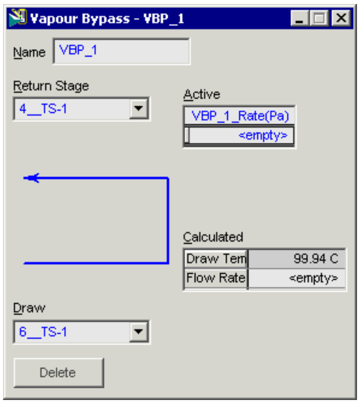
When installing a Pump Around, it is necessary to specify the draw stage, return stage, molar flow, and duty.

Vap Bypasses Page

As with the Pump Around, it is necessary to specify the draw and return stage, as well as the molar flow and duty for the vapour bypass. When you install the vapour bypass, the draw temperature and flowrate appear on the vapour bypass property view.

The vapour bypass flowrate is automatically added as a specification. The figure below shows the vapour bypass property view once the side operation has been installed.

Figure 2.45



Side Draws Page

The Side Draws page allows you to view, and edit information regarding the side draw streams in the column. The following is the information included on this page:

- Draw Stream
- Draw Stage
- Type (Vapour, Liquid or Water)
- Mole Flow
- Mass Flow
- Volume Flow

2.4.4 Rating Tab

The Rating tab has several pages, which are described in the table below.

Page	Description
Tray Sections	Provides information regarding tray sizing. On this page, you can specify the following: • Tray Section (Name) • Uniform Section. When this option is selected all tray stages have the same physical setup (diameter, tray type, and so forth). • Internal Type (tray type) • Tray Diameter • Tray Space • Tray Volume • Disable Heat Loss Calcs • Heat Model • Rating Calculations • **Hold Up (ft3)**. If you delete the weir height, you can then enter the hold up value, and the weir height is back-calculated. • Weeping Factor. The value is used to adjust the weeping in dynamic mode for low pressure drops.
Vessels	Provides information regarding vessel sizing. On this page, you can specify the following: • Vessel (Name) • Diameter • Length • Volume • Orientation • Vessel has a Boot • Boot Diameter • Boot Length • Hold Up (ft3)
Equipment	Contains a list of Other Equipment in the Column flowsheet.
Pressure Drop	Contains information regarding pressure drop across the column. On this page you can specify the following information: • Pressure Tolerance • Pressure Drop Tolerance • Damping Factor • Maximum Pressure Iterations • Top and Bottom column pressures

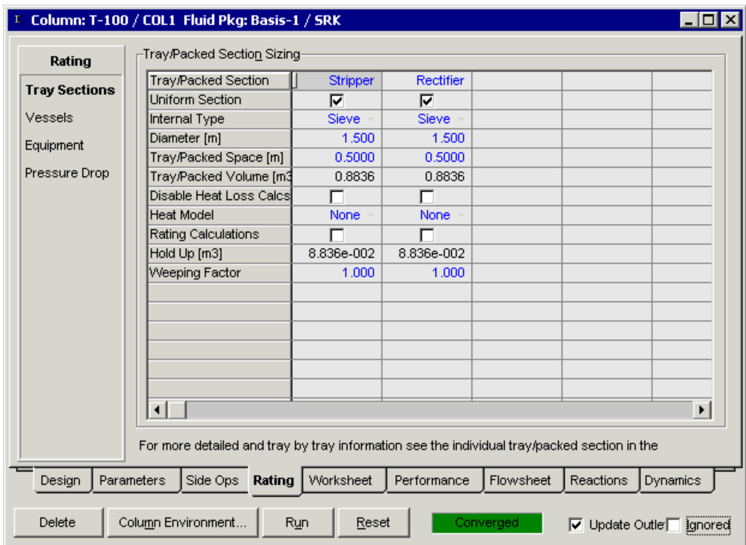
Tray Sections Page

The Tray Sections page contains all the required information for correctly sizing the column tray sections.

The required size information for the tray section can be calculated using the Tray Sizing utility.

The tray section diameter, weir length, weir height, and the tray spacing are required for an accurate and stable dynamic simulation. You must specify all the information on this page. With the exception of the tray volume, no other calculations are performed on this page.

Figure 2.46

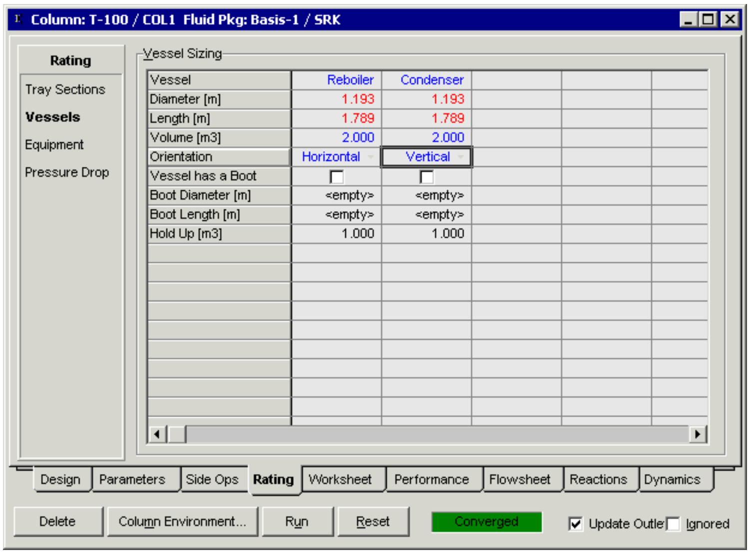


For multipass trays, simply enter the column diameter and the appropriate total weir length.

Vessels Page

The Vessels page contains the necessary sizing information for the different vessels in the column subflowsheet.

Figure 2.47



Equipment Page

The Equipment page contains a list of all the additional equipment, which is part of the column subflowsheet. The list does not contain equipment, which is part of the original template. Any extra equipment, which is added to the subflowsheet (pump arounds, side strippers, and so forth) is listed here. Double-clicking on the equipment name opens its property view on the Rating tab.

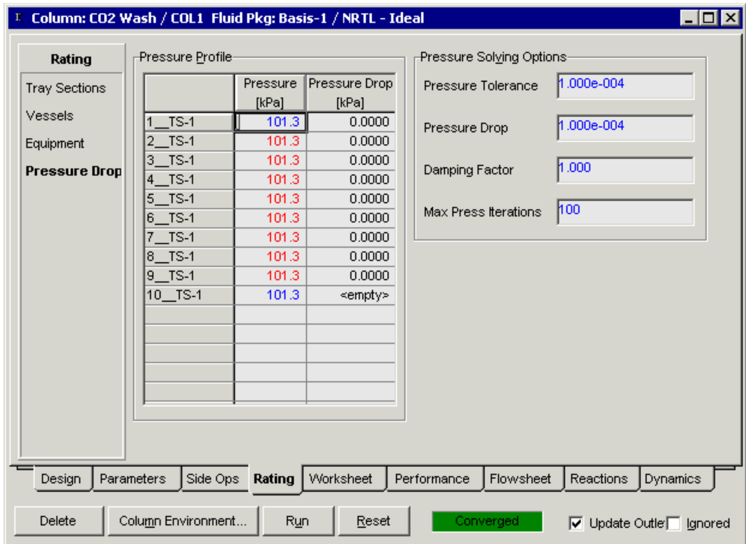
This page is not available in the Liquid-Liquid Extractor.

Pressure Drop Page

The Pressure Drop page allows you to specify the pressure drop across individual trays in the tray section. The pressure at each individual stage can also be specified. The Pressure Solving Options group allows you to adjust the following parameters:

- Pressure Tolerance
- Pressure Drop Tolerance
- Damping Factor
- Maximum Pressure Iterations

Figure 2.48



This page is not available in the Liquid-Liquid Extractor.

2.4.5 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the unit operation. The PF Specs page contains a summary of the stream property view's Dynamics tab.

The Column Environment also has its own Workbook.

2.4.6 Performance Tab

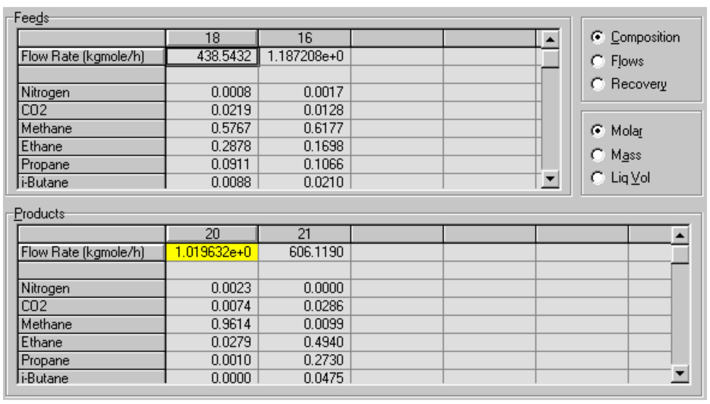
On the Performance tab, you can view the results of a converged column on the Summary page, Column Profiles page, and Feeds/Products page. You can also view the graphical and tabular presentation of the column profile on the Plots page.

You can view the results in molar, mass or liquid volume, by selecting the appropriate basis radio button.

Summary Page

The Summary page gives a tabular summary of the feed and product stream compositions, flows or the % recovery of the components in the product streams. When you select the Recovery radio button, the feed table displays the feed stream flowrate.

Figure 2.49



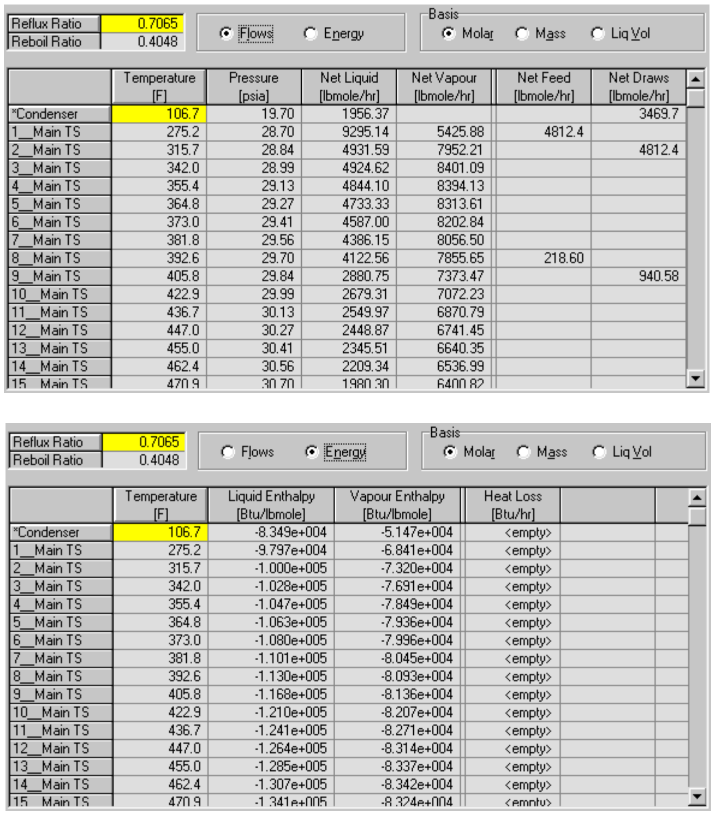
	18	16		
Flow Rate (kgmole/h)	438.5432	1.187208e+0		
Nitrogen	0.0008	0.0017		
CO2	0.0219	0.0128		
Methane	0.5767	0.6177		
Ethane	0.2878	0.1698		
Propane	0.0911	0.1066		
i-Butane	0.0088	0.0210		

	20	21			
Flow Rate (kgmole/h)	1.019632e+0	606.1190			
Nitrogen	0.0023	0.0000			
CO2	0.0074	0.0286			
Methane	0.9614	0.0099			
Ethane	0.0279	0.4940			
Propane	0.0010	0.2730			
i-Butane	0.0000	0.0475			

Column Profiles Page

The Column Profiles page gives a tabular summary of Column stage temperatures, pressures, flows, and duties.

Figure 2.50



Reflux Ratio	0.7065
Reboil Ratio	0.4048

	Temperature [F]	Pressure [psia]	Net Liquid [lbmole/hr]	Net Vapour [lbmole/hr]	Net Feed [lbmole/hr]	Net Draws [lbmole/hr]
*Condenser	106.7	19.70	1956.37			3469.7
1__Main TS	275.2	28.70	9295.14	5425.88	4812.4	
2__Main TS	315.7	28.84	4931.59	7952.21		4812.4
3__Main TS	342.0	28.99	4924.62	8401.09		
4__Main TS	355.4	29.13	4844.10	8394.13		
5__Main TS	364.8	29.27	4733.33	8313.61		
6__Main TS	373.0	29.41	4587.00	8202.84		
7__Main TS	381.8	29.56	4386.15	8056.50		
8__Main TS	392.6	29.70	4122.56	7855.65	218.60	
9__Main TS	405.8	29.84	2880.75	7373.47		940.58
10__Main TS	422.9	29.99	2679.31	7072.23		
11__Main TS	436.7	30.13	2549.97	6870.79		
12__Main TS	447.0	30.27	2448.87	6741.45		
13__Main TS	455.0	30.41	2345.51	6640.35		
14__Main TS	462.4	30.56	2209.34	6536.99		
15__Main TS	470.9	30.70	1980.30	6400.82		

Reflux Ratio	0.7065
Reboil Ratio	0.4048

	Temperature [F]	Liquid Enthalpy [Btu/lbmole]	Vapour Enthalpy [Btu/lbmole]	Heat Loss [Btu/hr]
*Condenser	106.7	-8.349e+004	-5.147e+004	<empty>
1__Main TS	275.2	-9.797e+004	-6.841e+004	<empty>
2__Main TS	315.7	-1.000e+005	-7.320e+004	<empty>
3__Main TS	342.0	-1.028e+005	-7.691e+004	<empty>
4__Main TS	355.4	-1.047e+005	-7.849e+004	<empty>
5__Main TS	364.8	-1.063e+005	-7.936e+004	<empty>
6__Main TS	373.0	-1.080e+005	-7.996e+004	<empty>
7__Main TS	381.8	-1.101e+005	-8.045e+004	<empty>
8__Main TS	392.6	-1.130e+005	-8.093e+004	<empty>
9__Main TS	405.8	-1.168e+005	-8.136e+004	<empty>
10__Main TS	422.9	-1.210e+005	-8.207e+004	<empty>
11__Main TS	436.7	-1.241e+005	-8.271e+004	<empty>
12__Main TS	447.0	-1.264e+005	-8.314e+004	<empty>
13__Main TS	455.0	-1.285e+005	-8.337e+004	<empty>
14__Main TS	462.4	-1.307e+005	-8.342e+004	<empty>
15__Main TS	470.9	-1.341e+005	-8.324e+004	<empty>

The liquid and vapour flows are net flows for each stage.

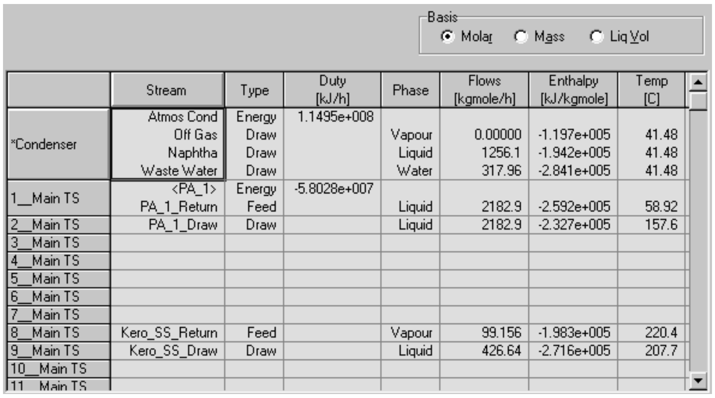
The Heat Loss column is empty unless you select a heat flow model in the column subflowsheet of Main TS property view on the Rating tab.

You can change the basis for which the data appears by selecting the appropriate radio button from the Basis group.

Feeds/Products Page

The Feeds/Products page gives a tabular summary of feed and product streams tray entry/exit, temperatures, pressures, flows, and duties.

Figure 2.51



Basis: Molar / Mass / Liq Vol

	Stream	Type	Duty [kJ/h]	Phase	Flows [kgmole/h]	Enthalpy [kJ/kgmole]	Temp [C]
*Condenser	Atmos Cond	Energy	1.1495e+008				
	Off Gas	Draw		Vapour	0.00000	-1.197e+005	41.48
	Naphtha	Draw		Liquid	1256.1	-1.942e+005	41.48
	Waste Water	Draw		Water	317.96	-2.841e+005	41.48
1__Main TS	<PA_1>	Energy	-5.8028e+007				
	PA_1_Return	Feed		Liquid	2182.9	-2.592e+005	58.92
2__Main TS	PA_1_Draw	Draw		Liquid	2182.9	-2.327e+005	157.6
3__Main TS							
4__Main TS							
5__Main TS							
6__Main TS							
7__Main TS							
8__Main TS	Kero_SS_Return	Feed		Vapour	99.156	-1.983e+005	220.4
9__Main TS	Kero_SS_Draw	Draw		Liquid	426.64	-2.716e+005	207.7
10__Main TS							
11__Main TS							

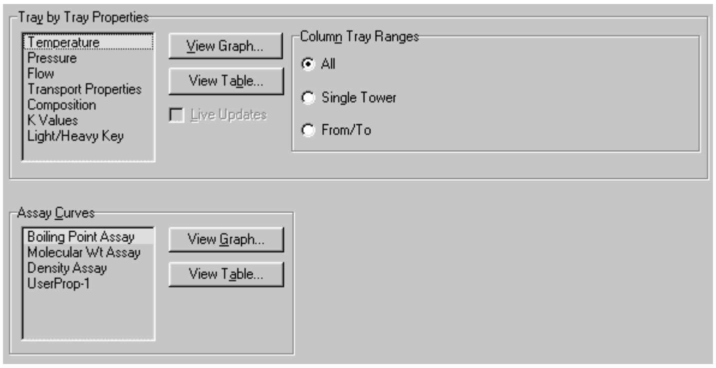
You can change the basis of the data by selecting the appropriate radio button from the Basis group. For the feeds and draw Streams, the VF column to the right of each flow value indicates whether the flow is vapour (V) or liquid (L). If the feed has been split, a star (*) follows the phase designation. If there is a duty stream on a stage, “Energy” appears in the Type column. The direction of the energy stream is indicated by the sign of the duty.

> **You can split a feed stream into its phase components either on the Setup page of the Flowsheet tab in the column property view or on the Options page of the Simulation tab in the Session Preferences property view.**

Plots Page

On the Plots page, you can view various column profiles or assay curves in a graphical or tabular format.

Figure 2.52



Select the **Live Updates** checkbox to update the profiles with every pass of the solver (in other words, a dynamic update). This checkbox is cleared by default, because the performance of the column can be a bit slower if the checkbox is selected and a profile is open.

Tray by Tray Properties Group

To view a column profile, follow this generalized procedure:

1. Select a profile from the list in the Tray by Tray Properties group.

   The choices include: Temperature, Pressure, Flow, Transport Properties, Composition, K Value, Light/Heavy Key, and Electrolyte Properties.

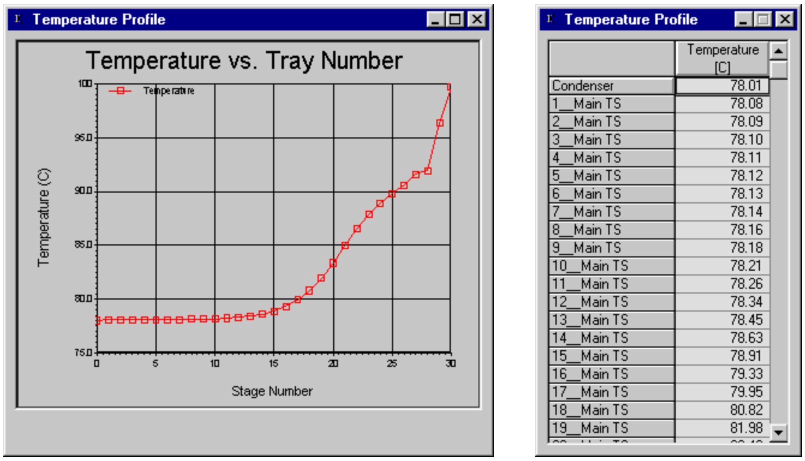
Electrolyte Properties are only available for cases with an electrolyte system.

2. In the Column Tray Ranges group, select the appropriate radio button:

Radio Button	Action
All	Displays the selected profile for all trays connected to the column (for example, main tray section, side strippers, condenser, reboiler, and so forth).
Single Tower	From the drop-down list, select a tray section. The main tray section along with the condenser and reboiler are considered one section, as is each side stripper.
From / To	Use the drop-down lists to specify a specific range of the column. The first field contains the tray that is located at a higher spot in the tower (for example, for top to bottom tray numbering, the first field could be tray 3 and the second tray 6).

3. After selecting a tray range, click either the **View Graph** button or the **View Table** button to display a plot or table respectively.

Figure 2.53



Refer to **Section 1.3.1 - Graph Control Property View** for more information.

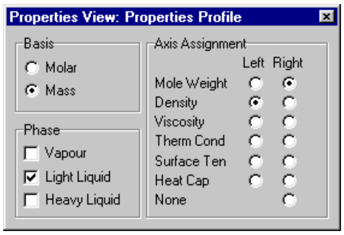
To make changes to the plot, right-click in the plot area, and select **Graph Control** from the object inspect menu.

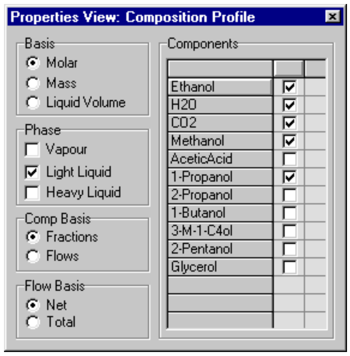
Plots and tables are expandible property views that can remain open without the column property view.

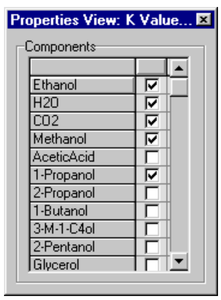
Depending on the profile selected, you have to make further specifications. For certain profiles, there is a Properties button on both the profile plot and table. By clicking this button, the Properties property view appears, where you can customize the display of your profile. Changes made on the Properties property view affect both the table and plot.

A description of the specifications available for each profile type are outlined in the following table.

Profile Type	Description
Temperature Profile	Displays the temperature for the tray range selected. No further specification is needed.
Pressure Profile	Displays the pressure of each tray in the selected range. No further specification is needed.
Flow Profile 	Displays the flow rate of each tray in the selected range. You can customise the data displayed using the Properties property view. In the Basis group, select molar, mass or liquid volume for your flow profile basis. In the Phase group, select the checkbox for the flow of each phase that you want to display. Multiple flows can be shown. If three phases are not present in the column, the Heavy Liquid checkbox is not available, and thus, the Light Liquid checkbox represents the liquid phase. In the Tray Flow Basis group, you can specify the stage tray flow basis by selecting the appropriate radio button: • Net. The net basis option only includes interstage flow. • Total. The total basis option includes draw and pump around flow.

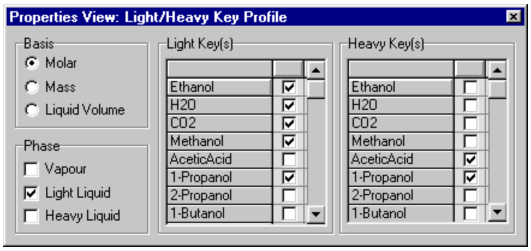
Profile Type	Description
Transport Properties Profile	Displays the selected properties from each tray in the selected range. You can customise the data displayed using the Properties property view:  In the Basis group, select molar or mass for the properties profile basis. In the Phase group, select the checkbox for the flow of each phase that you want to display on the graph. Multiple flows can be shown. If three phases are not present in the column, the Heavy Liquid checkbox is not available. The Properties Profile table displays **all** of the properties for the phase(s) selected. In the Axis Assignment group, by selecting a radio button under Left, you assign the values of the appropriate property to the left y-axis. To display a second property, choose the radio button under Right. The right y-axis then shows the range of the second property. If you want to display only one property on the plot, select the None radio button under Right.

<table>
<tr><th>Profile Type</th><th>Description</th></tr>
<tr><td>Composition</td><td>
Displays the selected component’s mole fraction of each tray in the selected range. You can customise the data displayed using the Properties property view.


In the Basis group, select molar, mass or liquid volume for the composition profile basis.
In the Phase group, select the checkbox for the flow of each phase that you want to display. Multiple flows can be shown. If the three phases are not present in the column, the Heavy Liquid checkbox is not available, and thus, the Light Liquid checkbox represents the liquid phase.
Choose either Fractions or Flows in the Comp Basis group by selecting the appropriate radio button.
The Components group displays a list of all the components that enter the tower. You can display the composition profile of any component by selecting the appropriate checkbox. The plot displays any combination of component profiles.
</td></tr>
<tr><td>K Values Profile</td><td>
Displays the K Values of each tray in the selected range. You can select which components you want included in the profile using the Properties property view.


</td></tr>
</table>

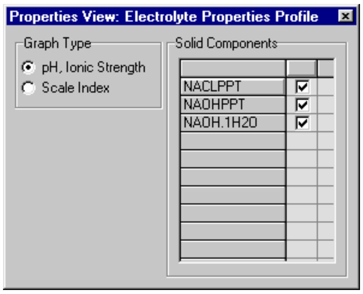
<table>
<tr><th>Profile Type</th><th>Description</th></tr>
<tr><td>Light / Heavy Key Profile</td><td>
Displays the fraction ratio for each stage. You can customise the data displayed using the Properties property view.



In the Basis group, select molar, mass or liquid volume for the profile basis.

In the Phase group, select Vapour, Light Liquid or Heavy Liquid for the profile phase.

In the Light Key(s) and Heavy Key(s) groups, you can select the key component(s) to include in your profile.
</td></tr>
<tr><td>Electrolyte Properties Profile</td><td>
Displays the pH and ionic strength or the scale index depending on which radio button you select in the Graph Type group.



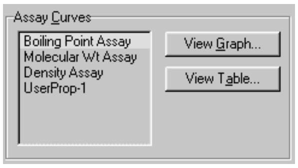
When you select the pH, Ionic Strength radio button, you can see how the pH value and ionic strength decrease or increase from tray to tray.

The Solid Components group displays a list of the solids that could form in the distillation column. You can select or clear the checkboxes to display or hide the scale tendency index value for the solid components in the table or graph.

The scale tendency index value refers to its tendency to form at the given tray conditions. Solids with a scale tendency index greater than 1 form, if the solid formation is governed by equilibrium (as oppose to kinetics), and if there are no other solids with a common cation or anion portion which also has scale tendency greater than 1.
</td></tr>
</table>

Assay Curves Group

Figure 2.54

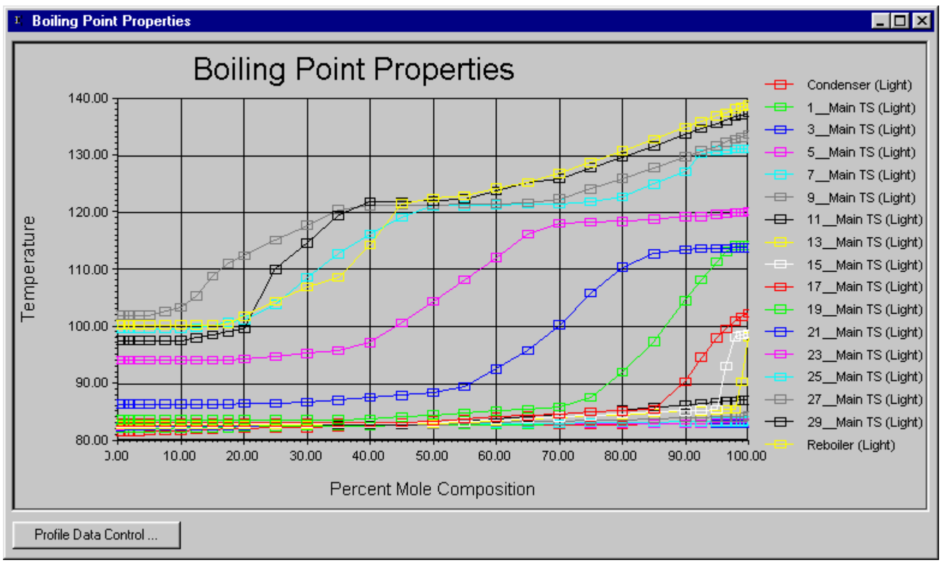


From the Assay Curves group, you can create plots and tables for the following properties:

- Boiling Point Assay
- Molecular Weight Assay
- Density Assay
- User Properties

For each of the options, you can display curves for a single tray or multiple trays. To display a plot or table, make a selection from the list, and click either the **View Graph** button or the **View Table** button. The figure below is an example of how a Boiling Point Properties plot appears.

Figure 2.55

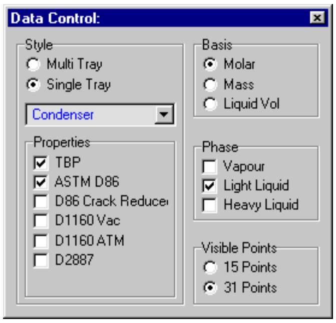


Data Control Property View

Click the Profile Data Control button, which is located on bottom left corner of every plot and table, to open the Data Control property view. This property view is common to all plots and tables on the Curves page. For a selected curve, all changes made on the Data Control property view affect the data of both the plot and table.

The Data Control property view consists of five groups as shown in the figure below.

Figure 2.56



The following table describes each data control option available according to group name.

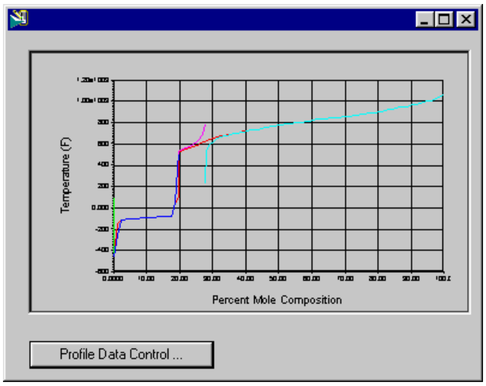
Group	Description
Style	Select either the Multi Tray or Single Tray radio button. The layout of the Data Control property view differs slightly for each selection. For the Single Tray selection, you must open the drop-down list and select one tray. If you select Multi Tray, the drop-down list is replaced by a list of all the trays in the column. Each tray has a corresponding checkbox, which you can select to display the tray property on the plot or table.
Properties	Displays the properties available for the plot or table. Each Curve option has its own distinct Properties group. For a single tray selection, you can choose as many of the boiling point curves as required. Select the checkbox for any of the following options: TBP, ASTM D86, D86 Crack Reduced, D1160 Vac, D1160 ATM, and D2887. When multiple trays have been chosen in the Style group, the checkbox list is replaced by a drop-down list. You can only choose one boiling point curve when displaying multiple trays.
Basis	Select molar, mass or liquid volume for the composition basis.
Phase	Select the checkbox for the flow of each phase that you want displayed. Multiple flows can be shown. If there are not three phases present in the column, the Heavy Liquid checkbox is not available, and thus, the Light Liquid checkbox represents the liquid phase.
Visible Points	The radio buttons in the Visible Points group apply to the plots only. Select either the 15 Points or 31 Points option to represent the number of data points which appear for each curve.

Refer to **Chapter 4 - HYSYS Oil Manager** in the **HYSYS Simulation Basis** guide for details on boiling point curves.

TBP Envelope Group

The TBP Envelope group contains only the **View Graph** button. You can click the **View Graph** button to display a TBP Envelope curve as shown in the figure below.

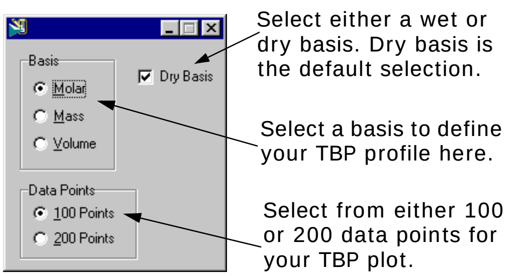
Figure 2.57



The curve allows you to view product stream distillation overlaid on the column feed distillation. This gives a visual representation of how sharp the separations are for each product. The sharpness of separation is adjusted using section and stripper efficiencies and front and back end shape factors.

Click the **Profile Data Control** button located on the property view above to open a property view for customizing your TBP Envelope curve.

Figure 2.58



2.4.7 Flowsheet Tab

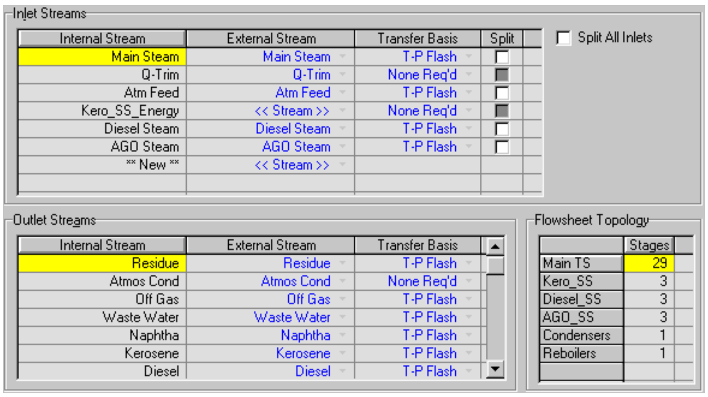
The Flowsheet tab contains the following pages:

- Setup
- Variables
- Internal Streams
- Mapping

Setup Page

The Setup page defines the connections between the internal (subflowsheet) and external (Parent) flowsheets.

Figure 2.59



To split all material inlet streams into their phase components before being fed to the column, select the **Split All Inlets** checkbox.

- If one of the material feed stream **Split** checkbox is clear, the **Split All Inlets** checkbox is cleared too.
- If you clear the **Split All Inlets** checkbox, none of the material inlet stream **Split** checkboxes are affected.

The Labels, as noted previously, attach the external flowsheet streams to the internal subflowsheet streams. They also perform the transfer (or translation) of stream information from the property package used in the parent flowsheet into the property package used in the Column subflowsheet (if the two property packages are different). The default transfer basis used for material streams is a P-H Flash.

The Transfer Basis is significant only when the subflowsheet and parent flowsheet Property Packages are different.

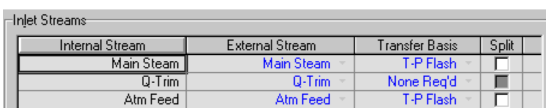
Flash Type	Action
T-P Flash	The pressure and temperature of the material stream are passed between flowsheets. A new vapour fraction is calculated.
VF-T Flash	The vapor fraction and temperature of the material stream are passed between flowsheets. A new Pressure is calculated.
VF-P Flash	The vapor fraction and pressure of the material stream are passed between flowsheets. A new temperature is calculated.
P-H Flash	The pressure and enthalpy of the material stream is passed between flowsheets. This is the default transfer basis.
User Specs	You can specify the transfer basis for a material Stream.
None Required	No calculation is required for an energy stream. The heat flow is simply passed between flowsheets.

When the **Split** checkbox for any of the inlet material streams is selected, the stream is split into its vapour and liquid phase components. The liquid stream is then fed to the specified tray, and the vapour phase to the tray immediately above the specified feed tray.

See the Summary page of the Performance tab to verify the split feed streams. An asterisk (*) following the phase indicator in the VF column indicates a split stream.

Energy streams and material streams connected to the top tray (condenser) cannot be split. The checkboxes for there variables appear greyed out.

Figure 2.60



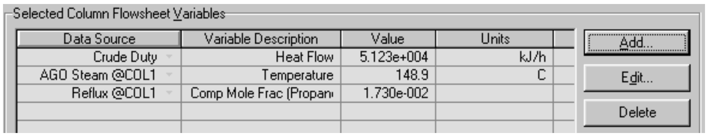
The Flowsheet Topology group provides stage information for each element in the flow sheet.

Flowsheet Variables Page (Main)

The Variables page allows you to select and monitor any flowsheet variables from one location. You can examine subflowsheet variables from the outside Column property view, without actually having to enter the Column subflowsheet environment.

You can add, edit or delete variables in the Selected Column flowsheet Variables group.

Figure 2.61



You can also use the Specifications page to view certain variables. Select the variable by adding a specification, and ensure that the Active and Estimate checkboxes are clear. The value of this variable appears in the Current value column, and this “pseudo-specification” do not affect the solution.

Adding a Variable

To add a variable in the Selected Column Flowsheet Variables group:

1. Click the **Add** button.
2. From the Variable Navigator, select each of the parameters for the variable.
3. Click the **OK** button.
4. The variable is added to the Selected Column Flowsheet Variables group.

Refer to **Section 1.3.9 - Variable Navigator Property View** for information on the Variable Navigator.

Editing a Variable

You can edit a variable in the Selected Column Flowsheet Variables group as follows:

1. Highlight a variable.
2. Click the **Edit** button.
3. Make changes to the selections in the Variable Navigator.
4. Click the **OK** button.

   If you decide that you do not want to keep the changes made in the variable navigator, click the **Cancel** button.

Deleting a Variable

You can remove a variable in any of the following ways:

- Select a variable, and click the **Delete** button.

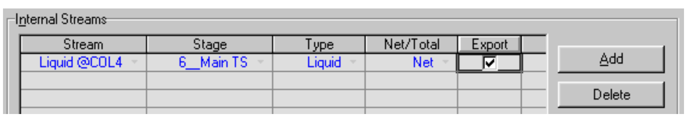
OR

- Select a variable, click the **Edit** button, and then click the **Disconnect** button on the Variable Navigator.

Internal Streams Page

On the Internal Streams page, you can create a flowsheet stream that represents any phase leaving any tray within the Column. Streams within operations attached to the main tray section (for example, side strippers, condenser, reboiler, and so forth) can also be targeted. Each time changes occur to the column, new information is automatically transferred to the stream which you have created.

Figure 2.62



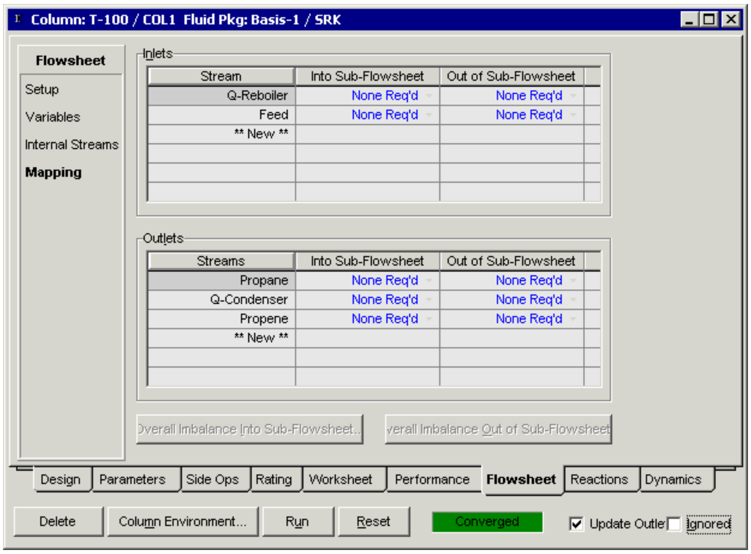
To demonstrate the addition of an internal stream, a stream representing the liquid phase flowing from tray 7 to tray 8 in the main tray section of a column is added:

1. Click the **Add** button.
2. In the Stream drop-down list, type the name of the stream named Liquid.
3. In the Stage drop-down list, select tray 6 or simply type 6, which locates the selection in the list.
4. In the Type drop-down list, select the phase that you want to represent. The options include Vapor, Liquid or Aqueous. Select Liquid in this case.
5. From the Net/Total drop-down list, select either Net or Total. For the stage 6 liquid, select Net.
    - Net represents the material flowing from the Stage you have selected to the next stage (above for vapour, below for liquid or aqueous) in the column.
    - Total represents all the material leaving the stage (for example, draws, pump around streams, and so forth).

Mapping Page

The Mapping page contains a table that displays the inlet and outlet streams from the column subflowsheet, and component maps for each boundary stream.

Figure 2.63



For more detail on the actual map collections and component maps themselves, refer to **Chapter 6 - Component Maps** in the **HYSYS Simulation Basis** guide.

If the fluid package of the column is the same as the main flowsheet, component maps are not needed (because components are the same on each side of the column boundary). None Req'd is the only option in the drop-down list of the **Into Sub-Flowsheet** and **Out of Sub-Flowsheet** columns. If the fluid packages are different, you can choose a map for each boundary stream. HYSYS lists appropriate maps based on the fluid package of each stream across the boundary.

Click the **Overall Imbalance Into Sub-Flowsheet** button or **Overall Imbalance Out of Sub-Flowsheet** button to view any mole, mass of liquid volume imbalance due to changes in fluid package. If there are no fluid package changes, then there are no imbalances.

2.4.8 Reactions Tab

This tab is not available for the Liquid-Liquid Extractor.

Reactive distillation has been used for many years to carry out chemical reactions, in particular esterification reactions. The advantages of using distillation columns for carrying out chemical reactions include:

- the possibility of driving the reaction to completion (break down of thermodynamic limitations for a reversible reaction), and separating the products of reactions in only one unit, thus eliminating recycle and reactor costs.
- the elimination of possible side reactions by continuous withdrawal of one of the products from the liquid phase.
- the operation at higher temperatures (boiling liquid), thus increasing the rate of reaction of endothermic reactions.
- the internal recovery of the heat of reaction for exothermic reactions, thereby replacing an equivalent amount of external heat input required for boil-up.

For any column in an electrolyte flowsheet, there is no option to add any reaction (reaction set) to the column. Conceptually, electrolyte thermo conducts a reactive and phase flash all together. HYSYS does not provide options to allow you to add external reactions to the unit operation.

The Reactions tab allows you to attach multiple reactions to the column. The tab consists of two pages:

- **Stages**. Allows you select the reaction set, and its scope across the column.
- **Results**. Displays the reaction results stage by stage.

Before adding a reaction to a column, you must first ensure that you are using the correct column Solving Method. HYSYS provides three solving methods which allow for reactive distillation.

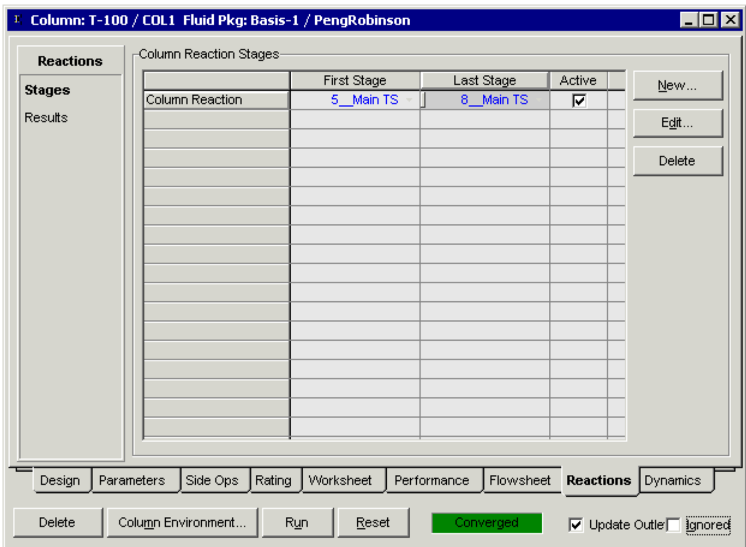
Solving Method	Reaction Type	Reaction Phase
Sparse Continuation Solver	Kinetic Rate, Simple Rate, Equilibrium Reaction	Vapor, Liquid
Newton Raphson Inside-Out	Kinetic Rate, Simple Rate	Liquid
Simultaneous Correction	Kinetic Rate, Simple Rate, Equilibrium Reaction	Vapor, Liquid, Combined Phase

The Sparse Continuation Solver method allows you to attach a reaction set to your column, which combines reaction types. Other solvers require that the attached reactions are of a single type.

Stages Page

The Stages page consists of the Column Reaction Stages group. The group contains the Column Reaction Stages table and three buttons.

Figure 2.64



Column Reaction Stages Table

The table consists of four columns, which are described in the table below.

Column	Description
Column Reaction Name	The name you have associated with the column reaction. This is not the name of the reaction set you set in the fluid package manager.
First Stage	The highest stage of the stage range over which the reaction is occurring.
Last Stage	The lowest stage of the stage range over which the reaction is occurring.
Active	Activates the associated reaction thereby enabling it to occur inside the column.

The property view also contains three buttons that control the addition, manipulation, and deletion of column reactions.

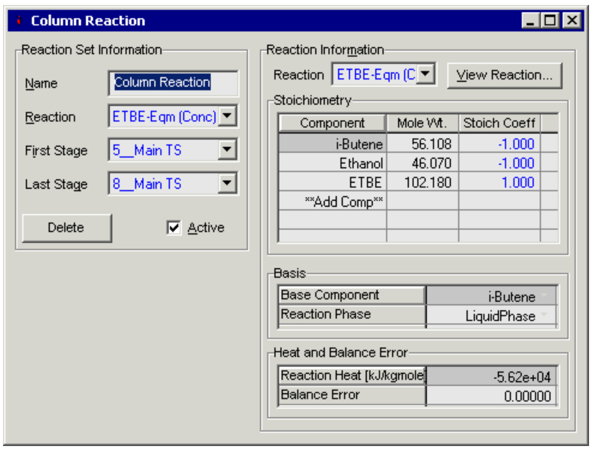
For more information of the Column Reaction property view, refer to the section on the **Column Reaction Property View**.

Button	Description
New	Allows you to add a new column reaction set via the Column Reaction property view.
Edit	Allows you to edit the column reaction set whose name is currently selected in Column Reaction Stages table. The selected reaction's Column Reaction property view appears.
Delete	Allows you to delete the column reaction set whose name is currently selected in the Column Reaction Stages table.

Column Reaction Property View

The Column Reaction property view allows you to add and revise column reactions.

Figure 2.65



The Column Reaction property view shown in the above figure consists of two groups:

- Reaction Set Information

  The Reaction Set Information group allows you to select the reaction set, and the scope of its application.
- Reaction Information

  The Reaction Information group contains thermodynamic and stoichiometric information about the reaction you are applying to the selected section of the column.

Reaction Set Information Group

The Reaction Set Information group consists of six objects:

Objects	Description
Name	The name you would like to associate with the column reaction. This is the name that appears in the **Column Reaction Name** column of the Column Reaction Stages table.
Reaction Set	Allows you to select a reaction set from a list of all the reaction sets attached to the fluid package.
First Stage	The upper limit for the reaction that is to occur over a range of stages.
Last Stage	The lower limit for the reaction that is to occur over a range of stages.
Delete	Deletes the Column Reaction from the column.
Active	Allows you to enable and disable the associated column reaction.

Reaction Information Group

The Reaction Information group contains the Reaction field, which allows you to select a reaction from the reaction set selected in the Reaction Set field.

Click the **View Reaction** button to open the selected reaction's Reaction property view. This group also contains three sub-groups, which allow you to view or specify the selected reactions properties.

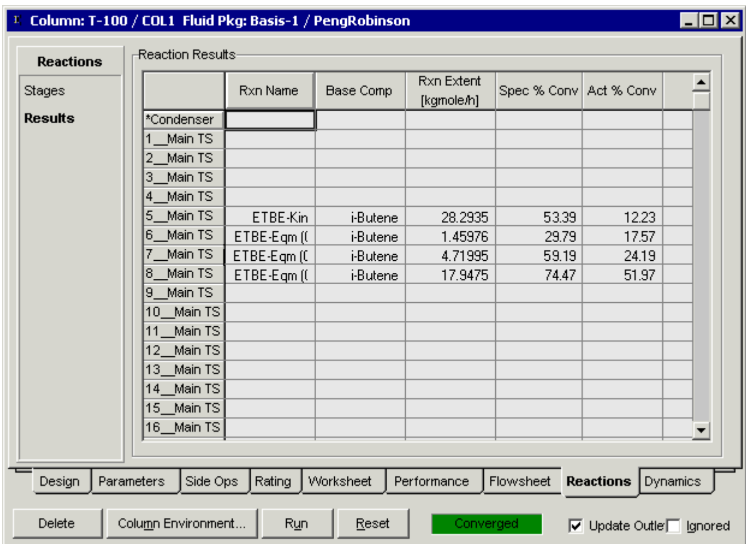
Sub-group	Description
Stoichiometry	Allows you to view and make changes to the stoichiometric formula of the reaction currently selected in the Reaction drop-down list. The group contains three columns: • **Components**. Displays the components involved in the reaction. • **Mole Wt.** Displays the molar weight of each component involved in the reaction. • **Stoich Coeff**. Stoichiometric coefficients associated with the reaction.

Sub-group	Description
Basis	Consists of two fields: • **Base Component.** Displays the reactant to which the reaction extent is calculated. This is often the limiting reactant. • **Reaction Phase**. Displays the phase for which the kinetic rate equations for different phases can be modeled in the same reactor. To see the possible reactions, click the Reaction Information button in the View Reaction group. You can make changes to the fields in these groups. These changes affect all the unit operations associated with this reaction. Click the **View Reactions** button for more information about the attached reaction.
Heat and Balance Error	Consists of two fields: • **Reaction Heat**. Displays the reaction heat. • **Balance Error**. Displays any error in the mass balance around the reaction.

Results Page

The Results page displays the results of a converged column.

Figure 2.66



The page consists of a table containing six columns. The columns are described in the following table:

Column	Description
1st column	Displays the name/number of the column stage.
Rxn Name	The name of the reaction occurring at this stage.
Base Comp	The name of the reactant component to which the calculated reaction extent is applied.
Rxn Extent	The consumption or production of the base component in the reaction.
Spec % Conv	Displays the percentage of conversion specified by you.
Act % Conv	Displays the percentage of conversion calculated by HYSYS.

If you have more than one reaction occurring at any particular stage, each reaction appears simultaneously.

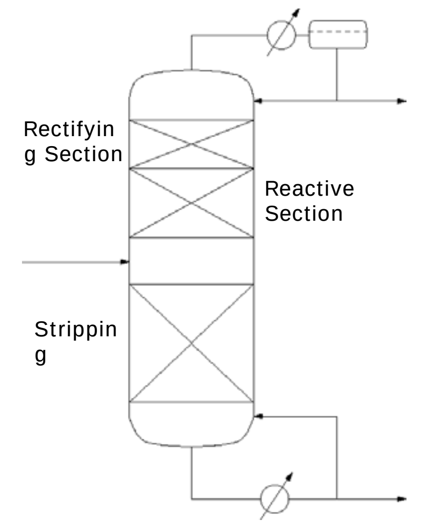
The Rxn Extent results appear only if the Sparse Continuation Solver is chosen as the Solving Method.

Design Tips for Reactive Distillation

[1]Although the column unit operations allows for multiple column reactions and numerous column configurations, a general column topography can be subdivided into three sections:

- Rectifying Section
- Reactive Section
- Stripping Section

Figure 2.67



While the Rectifying and Stripping Sections are similar to ordinary distillation, a reactive distillation column also has a Reactive Section. The Reactive Section of the column is where the main reactions occur. There is no particular requirement for separation in this section.

There are several unique operational considerations when designing a reactive distillation column:

- The operating pressure should be predicated on the indirect effects of pressure on reaction equilibrium.
- The optimum feed point to a reactive distillation column is *just below* the reactive section. Introducing a feed too far below the reactive section reduces the stripping potential of the column and results in increased energy consumption.
- Reflux has a dual purpose in reactive distillation. Increasing the reflux rate enhances separation and recycles unreacted reactants to the reaction zone thereby increasing conversion.
- Reboiler Duty is integral to reactive distillation as it must be set to ensure sufficient recycle of unreacted, heavy reactant to the reaction zone without excluding the light reactant from the reaction zone, if the reboiler duty is too high or too low, conversion, and purity can be compromised.

2.4.9 Dynamics Tab

The Dynamics tab contains the following pages:

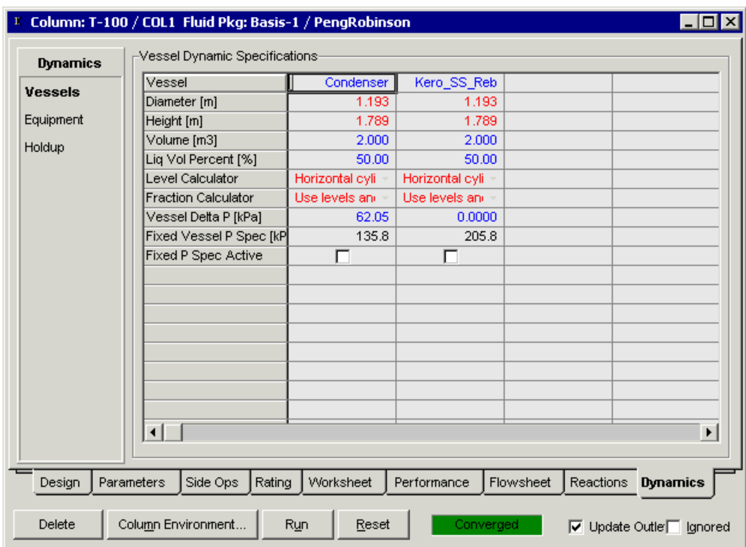
- Vessels
- Equipment
- Holdup

If you are working exclusively in Steady State mode or your version of HYSYS does not support dynamics, you are not required to change any information on the pages accessible through this tab.

Vessels Page

The Vessels page contains a summary of the sizing information for the different vessels contained in the column subflowsheet. In addition, it contains the possible dynamic specifications for these vessels.

Figure 2.68



Equipment Page

The Equipment page displays the same information as the Equipment page on the Rating tab. The difference is that double-clicking on the equipment name opens its property view on the Dynamics tab.

This page is not available for the Liquid-Liquid Extractor.

Holdup Page

The Holdup page contains a summary of the dynamic information calculated by HYSYS.

Column	Description
Pressure	Displays the calculated stage pressure.
Total Volume	Displays the stage volume.
Bulk Liq Volume	Displays the liquid volume occupying the stage.

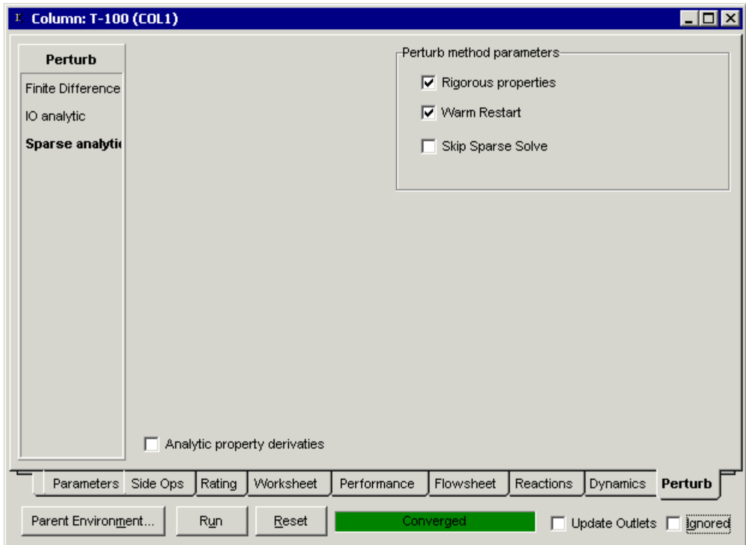
2.4.10 Perturb Tab

The Perturb tab is only available in the Column Runner property view. The Perturb tab allows you to control the way column solver calculates the partial derivatives. There are two types of independent controls.

Control	Description
Low Level Analytic	The Analytic property derivatives checkbox allows you to turn On and Off low level analytic derivatives support (in other words, derivatives of thermodynamic properties like Fugacity, Enthalpy, and Entropy by Temperature, Pressure, and Composition). At present this facility is available for Peng Robinson or Soave-Redlich-Kwong property packages in Sparse Continuation Solver context.
Optimizer Level Analytic	HYSYS Optimizer (RTO+) allows calculation of column analytic derivatives by stream Temperature, Pressure, Component Flow, Column Spec specified value, and Tear Variables.

The Sparse analytic page allows you to select a particular method of column analytic derivatives calculation.

Figure 2.69



The Perturb method parameters group provides tuning parameters for analytic column derivatives calculator.

- **Rigorous properties** checkbox. If active, rigorous thermodynamic properties are applied in Jacobi matrix calculation. If inactive, simple models (controlled by Control panel of Sparse solver) are applied instead for Enthalpy and Fugacity of thermodynamic phases. The last option may expedite derivative calculations.
- **Warm restart** checkbox. If active, additional Sparse linear solver information is preserved between Analytic derivative calculator calls (faster solution of linear system). If inactive, no Sparse linear solver information is stored (memory economy).
- **Skip Sparse Solve** checkbox. If active, Column solution phase is skipped (may allow faster execution).

2.5 Column Specification Types

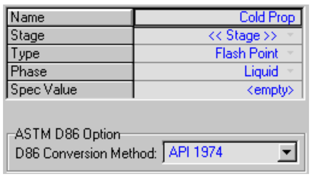
This section outlines the various Column specification (spec) types available along with relevant details. Specs are added and modified on the **Specs Page** or the **Monitor Page** of the Design tab.

Refer to the **Default Replaceable Specifications** in **Section 2.3.2 - Templates** for more information.

Adding and changing Column specifications is straightforward. If you have created a Column based on one of the templates, HYSYS already has default specifications in place. The type of default specification depends on which of the templates you have chosen.

2.5.1 Cold Property Specifications

Figure 2.70

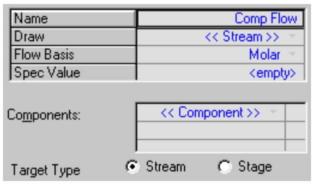


Cold Property	Description
Flash Point	Allows you to specify the Flash Point temperature (ASTM D93 flash point temperature closed cup) for the liquid or vapour flow on any stage in the column.
Pour Point	Allows you to specify the ASTM Pour Point temperature for the liquid or vapour flow on any stage in the column.
RON	Allows you to specify the Research Octane Number for the liquid or vapour flow on any stage.

2.5.2 Component Flow Rate

The flow rate (molar, mass or volume) of any component, or the total flow rate for any set of components, can be specified for the flow leaving any stage. If a side liquid or vapour draw is present on the selected stage, these are included with the internal vapour and liquid flows.

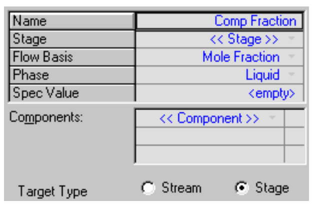
Figure 2.71



2.5.3 Component Fractions

The mole, mass or volume fraction can be specified in the liquid or vapour phase for any stage. You can specify a value for any individual component, or specify a value for the sum of the mole fractions of multiple components.

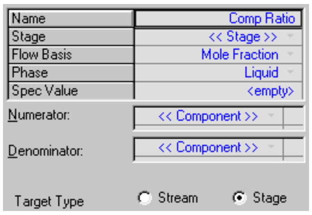
Figure 2.72



2.5.4 Component Ratio

The ratio (molar, mass or volume fraction) of any set of components over any other set of components can be specified for the liquid or vapour phase on any stage.

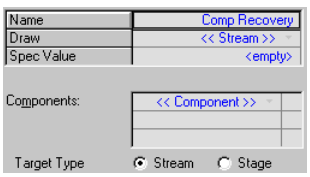
Figure 2.73



2.5.5 Component Recovery

Component recovery is the molar, mass or volume flow of a component (or group of components) in any internal or product stream draw divided by the flow of that component (or group) in the combined tower feeds. As the recovery is a ratio between two flows, you specify a fractional value. Also, there is no need to specify a Flow Basis since this is a ratio of the same component between specified stream and the combined tower feeds.

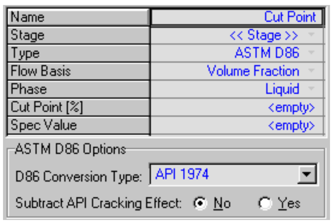
Figure 2.74



2.5.6 Cut Point

This option allows a cut point temperature to be specified for the liquid or vapour leaving any stage. The types are TBP, ASTM D86, D1160 Vac, D1160 ATM, and ASTM D2887. For D86, you are given the option to use ASTM Cracking Factor. For D1160, you are given an Atmospheric Pressure option. The cut point can be on a mole, mass or volume fraction basis, and any value from 0 to 100 percent is allowed.

Figure 2.75

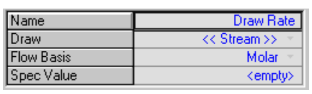


While initial and final cut points are permitted, it is often better to use 5 and 95 percent cut points to minimize the errors introduced at the extreme ends of boiling point curves.

2.5.7 Draw Rate

The molar, mass or volume flowrate of any product stream draw can be specified.

Figure 2.76



2.5.8 Delta T (Heater/Cooler)

The temperature difference across a Heater or Cooler unit operation can be specified. The Heater/Cooler unit must be installed in the Column subflowsheet, and the HYSIM Inside-Out, Modified HYSIM Inside-Out or Sparse Continuation solving methods must be selected on the Solver page of the Parameters tab.

2.5.9 Delta T (Streams)

The temperature difference between two Column subflowsheet streams can be specified.

Figure 2.77

Name	Delta Temp
First Stream	<< 1st Stream >>
Second Stream	<< 2nd Stream >>
Spec Value	<empty>

2.5.10 Duty

You can specify the duty for an energy stream.

Figure 2.78

Name	Duty
Energy Stream	<< Energy >>
Spec Value	<empty>

2.5.11 Duty Ratio

You can specify the duty ratio for any two energy streams. In addition to Column feed duties, the choice of energy streams also includes pump around duties (if available).

Figure 2.79

Name	Duty Ratio
Numerator Stream	<< Energy >>
Denominator Stream	<< Energy >>
Spec Value	<empty>

2.5.12 Feed Ratio

The Feed Ratio option allows you to establish a ratio between the flow rate on or from any stage in the column, and the external feed to a stage. You are prompted for the stage, flow type (Vapor, Liquid, Draw), and the external feed stage.

Figure 2.80

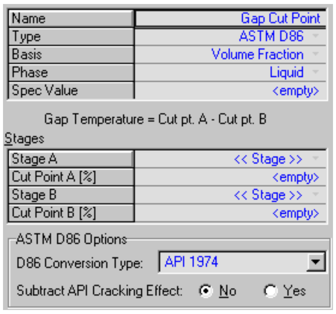
Name	Feed Ratio
Flow Type	Tray Vapour
Stage	<< Stage >>
Feed Stage	<< Stage >>
Flow Basis	Molar
Spec Value	<empty>

This type of specification is useful for turn down or overflash of a crude feed.

2.5.13 Gap Cut Point

The Gap Cut Point is defined as the temperature difference between a cut point (Cut Point A) for the liquid or vapour leaving one stage, and a cut point (Cut Point B) on a different stage.

Figure 2.81



This specification is best used in combination with at least one flow specification; using this specification with a Temperature specification can produce non-unique solutions.

You have a choice of specifying the distillation curve to be used:

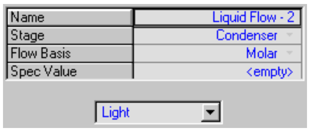
- TBP
- ASTM D86
- D1160 Vac
- D1160 ATM
- ASTM D2887

You can define Cut Point A and Cut Point B, which together must total 100%. The cut points can be on a mole, mass or volume basis.

2.5.14 Liquid Flow

The net molar, mass or volume liquid (Light or Heavy) flow can be specified for any stage.

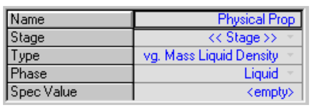
Figure 2.82



2.5.15 Physical Property Specifications

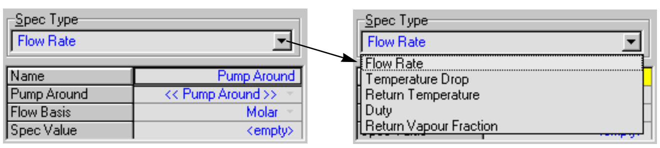
The mass density can be specified for the liquid or vapour on any stage.

Figure 2.83



2.5.16 Pump Around Specifications

Figure 2.84



Specification	Description
Flow Rate	The flow rate of the Pump Around can be specified in molar, mass, or liquid volume units.
Temperature Drop	Allows you to specify the temperature drop across a Pump Around exchanger. The conditions for using this specification are the same as that stated for the Pump Around return temperature.
Return Temperature	The return temperature of a Pump Around stream can be specified. Ensure that you have not also specified both the pump around rate and the duty. This would result in the three associated variables (flow rate, side exchanger duty, and temperature) all specified, leaving HYSYS with nothing to vary in search of a converged solution.
Duty	You can specify the duty for any Pump Around.
Return Vapor Fraction	You can specify the return vapour fraction for any Pump Around.
Duty Ratio	To specify a Pump Around duty ratio for a Column specification, add a Column Duty Ratio spec instead, and select the Pump Around energy streams to define the duty ratio.

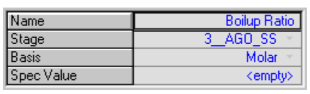
Refer to **Section 2.5.11 - Duty Ratio** for more information.

The Pump Around Rate, as well as the Pump Around Temperature Drop are the default specifications HYSYS requests when a pump around is added to the column.

2.5.17 Reboil Ratio

You can specify the molar, mass or volume ratio of the vapour leaving a specific stage to the liquid leaving that stage.

Figure 2.85

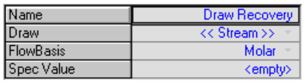


2.5.18 Recovery

The Recovery spec is the recovery of the total feed flow in the defined outlet streams (value range between 0 and 1).

$$\frac{\text{molar flow of draw stream}}{\text{total molar feed flow}} = \text{\% recovery} \tag{2.7}$$

Figure 2.86

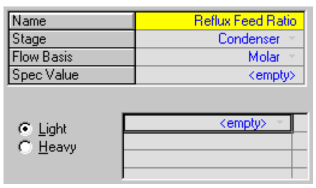


2.5.19 Reflux Feed Ratio

The Reflux Feed Ratio spec is the fraction of the reflux flow divided by the reference flow for the specified stage and phase.

$$\frac{\text{reflux flow}}{\text{reference flow}} = \text{reflux feed ratio} \tag{2.8}$$

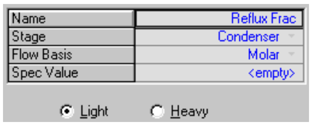
Figure 2.87



2.5.20 Reflux Fraction Ratio

The Reflux Fraction Ratio spec is the fraction or % of liquid that is being refluxed on the specified stage (value range between 0 and 1).

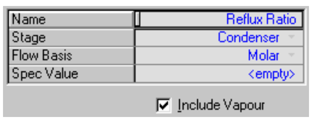
Figure 2.88



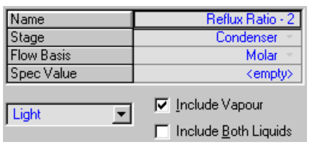
2.5.21 Reflux Ratio

The Reflux Ratio is the molar, mass or volume flow of liquid (Light or Heavy) leaving a stage, divided by the sum of the vapour flow from the stage plus any side liquid flow.

Figure 2.89



Reflux Ratio property view for general column



Reflux Ratio property view for three phase distillation column

The Reflux Ratio specification is normally used only for top stage condensers, but it can be specified for any stage. For a Partial Condenser:

- Selecting the **Include Vapour** checkbox, gives the following equation for the reflux ratio:

$$\text{Reflux Ratio} = \frac{R}{V+D}$$

- Clearing the **Include Vapour** checkbox, gives the following equation for the reflux ratio:

$$\text{Reflux Ratio} = \frac{R}{D}$$

where:

R = liquid reflux to column

V = vapor product

D = distillate product

2.5.22 Tee Split Fraction

The split fraction for a Tee operation product stream can be specified. The Tee must be installed within the Column subflowsheet and directly attached to the column, for example, to a draw stream, in a pump around circuit, and so forth. Also, the Modified HYSIM Inside-Out solving method must be selected.

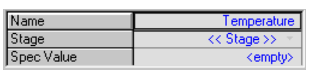
Refer to **Section 7.5 - Tee** for details on the Tee operation.

Tee split fraction specifications are automatically installed as you install the tee operation in the Column subflowsheet; however, you can select which specifications become active on the Monitor page or Specs page. Changes made to the split fraction specification value are updated on the Splits page of the tee operation.

2.5.23 Tray Temperature

The temperature of any stage can be specified.

Figure 2.90

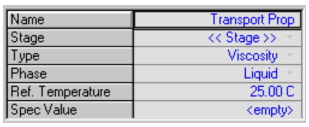


2.5.24 Transport Property Specifications

The viscosity, surface tension or thermal conductivity can be specified for the liquid leaving any stage. The viscosity or thermal conductivity can be specified for the vapour leaving any stage. A reference temperature must also be given.

The computing time required to satisfy a vapour viscosity specification can be considerably longer than that needed to meet a liquid viscosity specification.

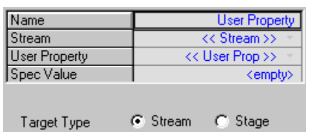
Figure 2.91



2.5.25 User Property

A User Property value can be specified for the flow leaving any stage. You can choose any installed user property in the flowsheet, and specify its value. The basis used in the installation of the user property is used in the spec calculations.

Figure 2.92



2.5.26 Vapor Flow

The net molar, mass or volume vapor flow can be specified for any stage. Feeds and draws to that tray are taken into account.

Figure 2.93

Name	Vapour Flow
Stage	<< Stage >>
Flow Basis	Molar
Spec Value	<empty>

2.5.27 Vapor Fraction

The vapour fraction of a stream exiting a stage can be specified.

Figure 2.94

Name	Vapour Fraction
Stage	3__AGO_SS
Spec Value	<empty>

2.5.28 Vapor Pressure Specifications

Two types of vapour pressure specifications are available:

- true vapour pressure (@100°F)
- Reid vapour pressure.

Figure 2.95

Name	Vapour Press
Stage	<< Stage >>
Type	Vap Press (100F)
Phase	Liquid
Spec Value	<empty>

Vapor Type	Description
Vapor Pressure	The true vapour pressure at 100°F can be specified for the vapour or liquid leaving any stage.
Reid Vapor Pressure	Reid vapour pressure can be specified for the vapour or liquid leaving any stage. The specification must always be given in absolute pressure units.

2.6 Column Stream Specifications

Column stream specifications must be created in the Column subflowsheet. Unlike other specifications, the stream specification is created through the stream's property view, and not the Column Runner Specs page. To be able to add a specification to a stream:

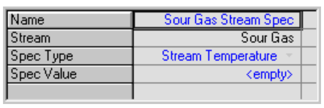
- The Modified HYSIM Inside-out solving method must be chosen for the solver.

Only one stream specification can be created per draw stream.

- The stream must be a draw stream.

The Create Column Stream Spec button on the Conditions page of the Worksheet tab is available only on Stream property views within the Column subflowsheet. When you click on the Create Column Stream Spec button, the Stream Spec property view appears.

Figure 2.96



- For draw streams from a separation stage (tray section stage, condenser or reboiler) only a stream temperature specification can be set.

- For a non-separation stage streams (from pumps, heaters, and so forth) either a temperature or a vapour fraction specification can be set.
- For any given stage, only one draw stream specification can be active at any given time.

Creating a new stream specification for a stage, or activating a specification automatically deactivates all other existing draw stream specifications for that stage.

Once a specification is added for a stream, the button on the Conditions page of the Worksheet tab changes from Create Column Stream Spec to View Column Stream Spec, and can be clicked to view the Stream Specification property view.

You can only add Column Stream Specifications via the Stream property view of a draw stream within the Column subflowsheet.

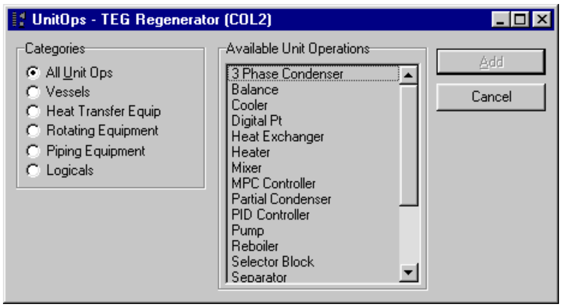
2.7 Column-Specific Operations

Refer to **Section 1.2.1 - Installing Operations** for more information.

The procedure for installing unit operations in a Column subflowsheet is the same as in the main flowsheet.

The UnitOps property view for the Column appears by selecting the **Flowsheet | Add Operation** command from the menu bar, or by pressing **F12**.

Figure 2.97



The unit operations available within the Column subflowsheet are listed in the following table.

Operation Category	Types
Vessels	3-Phase Condenser, Partial Condenser, Reboiler, Separator, Total Condenser, Tray Section
Heat Transfer Equipment	Cooler, Heater, Heat Exchanger
Rotating Equipment	Pump
Piping Equipment	Valve
Logicals	Balance, Digital Pt, PID Controller, Selector Block, Transfer Function Block

Most operations shown here are identical to those available in the main flowsheet in terms of specified and calculated information, property view structure, and so forth.

Only the operations which are applicable to a Column operations are available within the Column subflowsheet.

There are also additional unit operations which are not available in the main flowsheet. They are:

- Condenser (Partial, Total, 3-Phase)
- Reboiler
- Tray Section

The Bypasses and Side Operations (side strippers, pump arounds, and so forth) are available on the Side Ops page of the Column property view.

Refer to **Section 7.24.4 - Access Column or Subflowsheet PFDs** in the **HYSYS User Guide** for more information

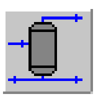
You can open a property view of the Column PFD from the main build environment. This PFD only provides you with the ability to modify stream and operation parameters. You cannot add and delete operations or break stream connections. These tasks can only be performed in the Column subflowsheet environment.

2.7.1 Condenser

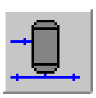
The Condenser is used to condense vapour by removing its latent heat with a coolant. In HYSYS, the condenser is used only in the Column Environment, and is generally associated with a Column Tray Section.

There are four types of Condensers:

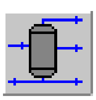
Condenser Type	Description
Partial	Feed is partially condensed; there are vapour and liquid product streams. The Partial Condenser can be operated as a total condenser by specifying the vapour stream to have zero flowrate. The Partial Condenser can be used as a Total Condenser simply by specifying the vapour flowrate to be zero.
Total	Feed is completely condensed; there is a liquid product only.
Three-Phase - Chemical	There are two liquid product streams and one vapour product stream.
Three-Phase - Hydrocarbon	There is a liquid product streams and a water product stream and one vapour product stream.



Partial Condenser icon



Total Condenser icon



Three-Phase Condenser icon

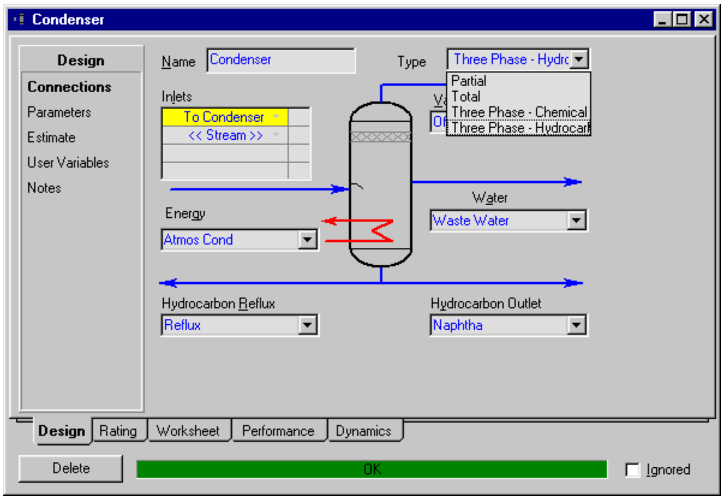
When you add a Column to the simulation using a pre-defined template, there can be a condenser attached to the tower (for example, in the case of a Distillation Column).

To manually add a Condenser do one of the following:

- In the Column environment, press **F12** and make the appropriate selection from the UnitOps property view.
- In the Column environment, press **F4** and click a Condenser icon from the Column Palette.

The Condenser property view uses a **Type** drop-down list, which allows you to switch between condenser types without having to delete and re-install a new piece of equipment.

Figure 2.98



When you switch between the condenser types, the pages change appropriately. For example, the **Connections** page for the Total Condenser does not show the vapor stream. If you switch from the Partial to Total Condenser, the vapor stream is disconnected. If you then switch back, you have to reconnect the stream.

The Condenser property view has the same basic five tabs that are available on any unit operation:

- Design
- Rating
- Worksheet
- Performance
- Dynamics

It is necessary to specify the connections and the parameters for the Condenser. The information on the Dynamics tab are not relevant in steady state.

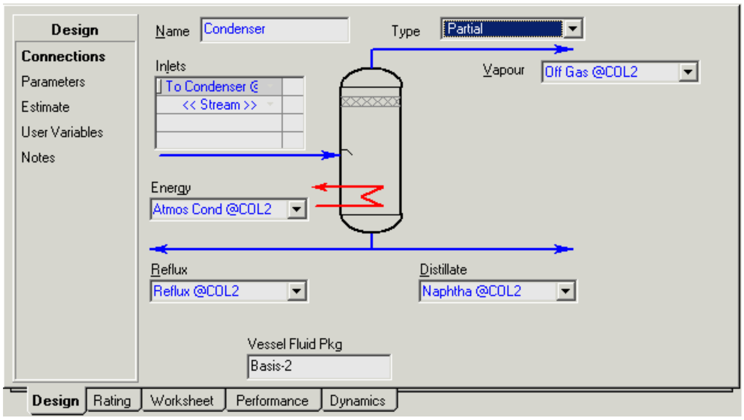
Design Tab

The Design tab contains options to configure the Condenser.

Connections Page

On the Connections page, you can specify the operation name, as well as the feed(s), vapour, water, reflux, product, and energy streams.

Figure 2.99

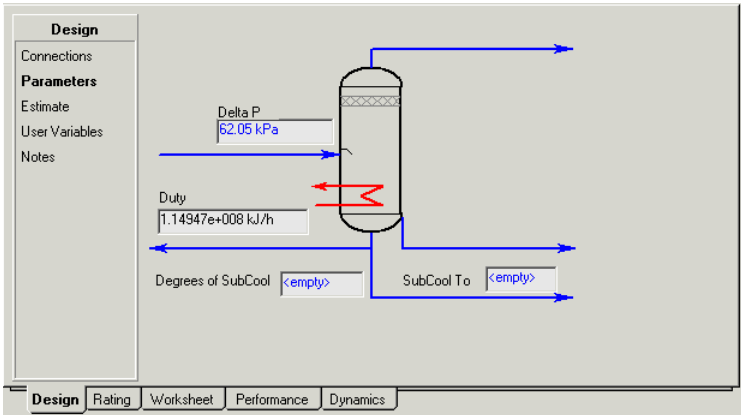


The Connections page shows only the product streams, which are appropriate for the selected condenser. For example, the Total Condenser does not have a vapour stream, as the entire feed is liquefied. Neither the Partial nor the Total Condenser has a water stream.

The Condenser is typically used with a tray section, where the vapour from the top tray of the column is the feed to the condenser, and the reflux from the condenser is returned to the top tray of the column.

Parameters Page

Figure 2.100



The condenser parameters that can be specified are:

- Pressure Drop
- Duty

It is better to use a duty spec than specifying the heat flow of the duty stream.

- Subcooling Data

Pressure Drop

The Pressure Drop across the condenser (Delta P) is zero by default. It is defined in the following expression:

$$P = P_v = P_l = P_{feed} - \Delta P \quad (2.9)$$

where:

P = *vessel pressure*

P_v = *pressure of vapour product stream*

P_l = *pressure of liquid product stream*

P_{feed} = *pressure of feed stream to condenser*

ΔP = *pressure drop in vessel (Delta P)*

You typically specify a pressure for the condenser during the column setup, in which case the pressure of the top stage is the calculated value.

Duty

The Duty for the energy stream can be specified here, but this is better done as a column spec (defined on the Monitor page or Specs page of the Column property view). This allows for more flexibility when adjusting specifications, and also introduces a tolerance.

If you specify the duty, it is equivalent to installing a duty spec, and a degree of freedom is used.

The Duty should be positive, indicating that energy is being removed from the Condenser feed.

The steady state condenser energy balance is defined as:

$$H_{feed} - Duty = H_{vapour} + H_{liquid} \tag{2.10}$$

where:

H_{feed} = *heat flow of the feed stream to the condenser*

H_{vapour} = *heat flow of the vapour product stream*

H_{liquid} = *heat flow of the liquid product stream(s)*

SubCooling

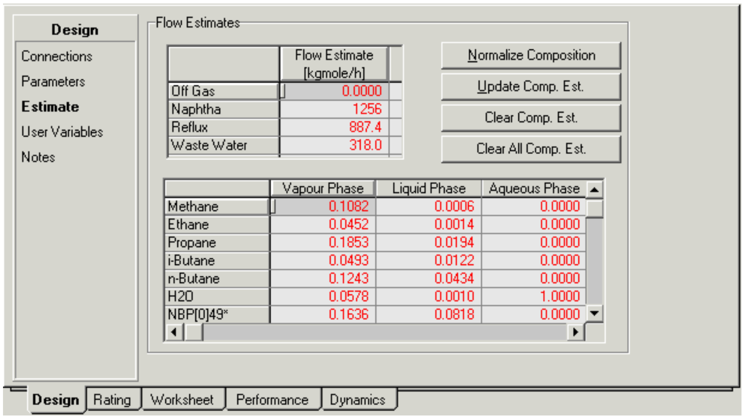
In some instances, you want to specify Condenser SubCooling. In this situation, either the Degrees of SubCooling or the SubCooled Temperature can be specified. If one of these fields is set, the other is calculated automatically.

In steady state, SubCooling applies only to the Total Condenser. There is no SubCooling in dynamics.

Estimate Page

On the Estimate page you can estimate the flows and phase compositions of the streams exiting the Condenser.

Figure 2.101



You can enter any value for fractional compositions, and click the Normalize Composition button to have HYSYS normalize the values such that the total equals 1. This button is useful when many components are available, but you want to specify compositions for only a few. HYSYS also specifies any <empty> compositions as zero.

HYSYS re-calculates the phase composition estimates when you click the Update Comp. Est. button. Clicking this button also removes any of the estimated values you entered for the phase composition estimates.

Click the Clear Comp. Est. button to clear the phase compositions estimated by HYSYS. This button does not remove any estimate values you entered. You can clear the all estimate values by clicking the Clear All Comp. Est. button.

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

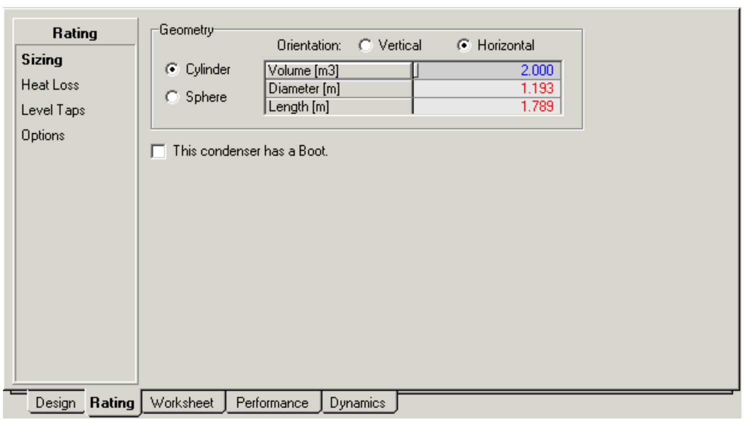
Rating Tab

The Rating tab contains options that are applicable in both Steady State and Dynamics mode.

Sizing Page

The Sizing page contains all the required information for correctly sizing the condenser.

Figure 2.102



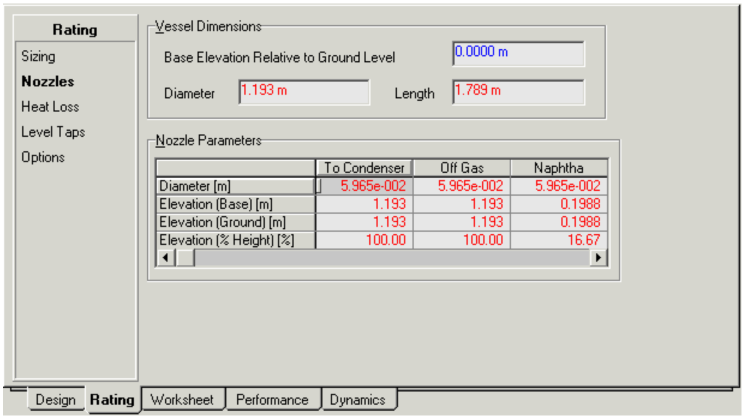
You can select either vertical or horizontal orientation, and cylinder or sphere. You can either enter the volume or dimensions for your condenser. You can also indicate whether or not the condenser has a boot associated with it. If it does, then you can specify the boot dimensions.

Nozzles Page

Refer to **Section 1.3.6 - Nozzles Page** for more information.

The Nozzles page contains information regarding the elevation and diameter of the nozzles. The information provided in the Nozzles page is applicable only in Dynamic mode.

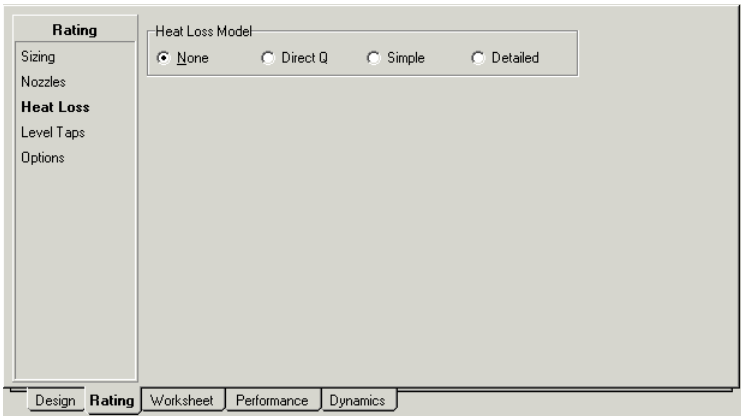
Figure 2.103



Heat Loss Page

The Heat Loss page allows you to specify the heat loss from individual trays in the tray section. You can choose either a Direct Q, Simple, or Detailed heat loss model or no heat loss from the Heat Loss Mode group.

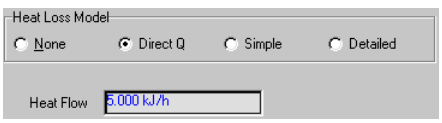
Figure 2.104



Direct Q Heat Loss Model

The Direct Q model allows you to either specify the heat loss directly, or have the heat loss calculated from the Heat Flow for the condenser.

Figure 2.105

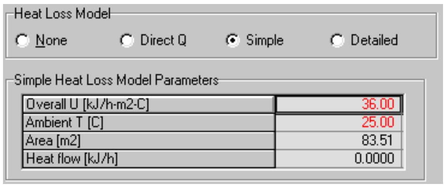


Simple Heat Loss Model

The Simple model allows you to calculate the heat loss from these specified values:

- Overall U value
- Ambient Temperature

Figure 2.106

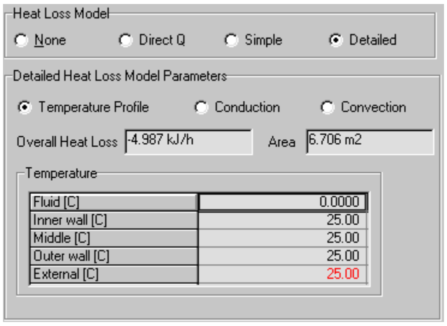


Detailed Heat Loss Model

Refer to **Section 1.6.1 - Detailed Heat Model** in the **HYSYS Dynamic Modeling** guide for more information.

The Detailed model allows you to specify more detailed heat transfer parameters.

Figure 2.107



Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the Condenser.

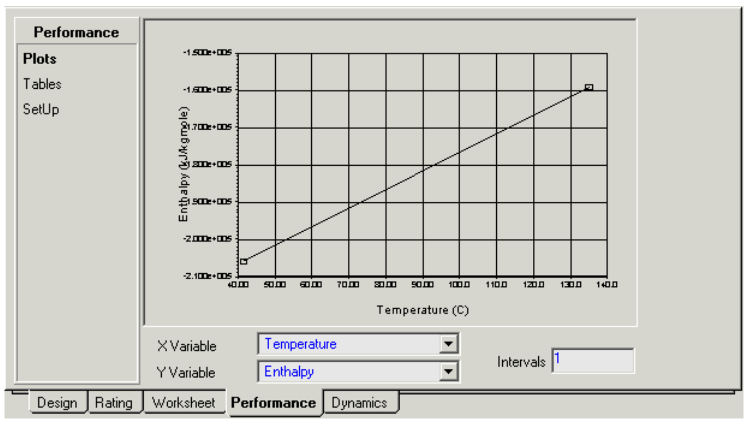
The PF Specs page is relevant to dynamics cases only.

Performance Tab

The Performance tab has the following pages:

- Plots

Figure 2.108



In steady state, the displayed plots are all straight lines. Only in Dynamic mode, when the concept of zones is applicable, do the plots show variance across the vessels.

- Tables
- SetUp

From these pages you can select the type of variables you want to calculate and plot, view the calculated values, and plot any combination of the selected variables. The default selected variables are temperature, pressure, heat flow, enthalpy, and vapor fraction. At the bottom of the Plots or Tables page, you can specify the interval size over which the values should be calculated and plotted.

Dynamics Tab

The Dynamics tab contains the following pages:

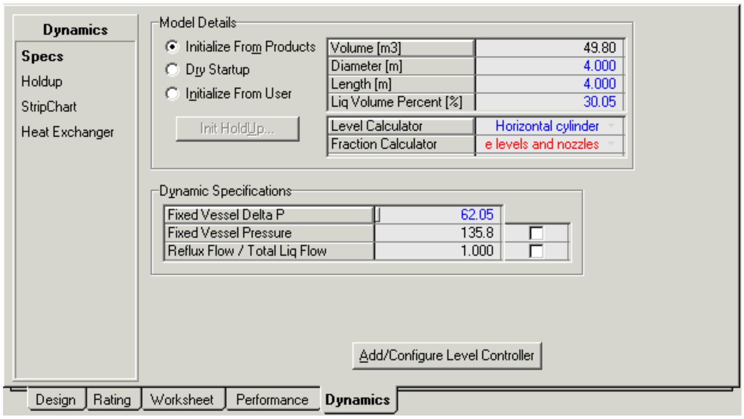
- Specs
- Holdup
- StripChart
- Heat Exchanger

You are not required to modify information on the Dynamics tab when working in Steady State mode.

Specs Page

The Specs page contains information regarding initialization modes, condenser geometry, and condenser dynamic specifications.

Figure 2.109



Model Details

In the Model Details group, you can specify the initial composition and amount of liquid that the separator should start with when you start dynamics. This is done via the initialization mode which is discussed in the table below.

Initialization Mode	Description
Initialize from Products	The composition of the holdup is calculated from a weighted average of all products exiting the holdup. A PT flash is performed to determine other holdup conditions. The liquid level is set to the value indicated in the Liq Volume Percent field.
Dry Startup	The composition of the holdup is calculated from a weighted average of all feeds entering the holdup. A PT flash is performed to determine other holdup conditions. The liquid level in the Liq Volume Percent field is set to zero.
Initialize from User	The composition of the liquid holdup in the condenser is user specified. The molar composition of the liquid holdup can be specified by clicking the Init Holdup button. The liquid level is set to the value indicated in the Liq Volume Percent field.

The Initialization Mode can be changed any time when the integrator is not running. The changes cause the vessel to re-initialize when the integrator is started again.

The condenser geometry can be specified in the Model Details group. The following condenser geometry parameters can be specified in the same manner as the Geometry group on Sizing page of the Rating tab:

- Volume
- Diameter
- Height (Length)
- Geometry (Level Calculator)

The Liquid Volume Percent value is also displayed in this group. You can modify the level in the condenser at any time. HYSYS then uses that level as an initial value when the integrator is run.

Refer to the section on the **Nozzles Page** for more details.

The Fraction Calculator determines how the level in the condenser and the elevation and diameter of the nozzle affects the product composition. There is only one Fraction Calculation mode available, it is called Use Levels and Nozzles. The calculations are based on how the nozzle location and vessel liquid level affect the product composition.

Dynamic Specifications

The Dynamic Specifications group contains fields, where you can specify what happens to the pressure and reflux ratio of the condenser when you enter dynamic mode.

The Fixed Pressure Delta P field allows you to impose a fixed pressure drop between the vessel and all of the feed streams. This is mostly supported for compatibility with Steady State mode. In Dynamic mode, you are advised to properly account for all pressure losses by using the appropriate equipment such as valves or pumps or static head contributions. A zero pressure drop should preferably be used here otherwise you may get unrealistic results such as material flowing from a low to a high pressure area.

The Fixed Vessel Pressure field allows you to fix the vessel pressure in Dynamic mode. This option can be used in simpler models where you do not want to configure pressure controllers and others, or if the vessel is open to the atmosphere. In general the specification should not be used, because the pressure should be determined by the surrounding equipment.

The Reflux Flow/Total Liquid Flow field provides you with a simple reflux ratio control option, and the ratio determines the reflux flow rate divided by the sum of the reflux and distillate flow rates.

This option allows you to set up simple models without having to add the valves, pumps, and controller that would normally be present. This option does not always give desirable results under all conditions such as very low levels or reversal of some of the streams.

The Add/Configure Level Controller button installs a level controller on the distillate (liquid) outlet stream if one is not already present. If this stream has a valve immediately downstream of the vessel, the controller is configured to control the valve rather than the stream directly. In any case, the controller is configured with some basic tuning parameters, but you can adjust those.

The default tuning values are as follows:

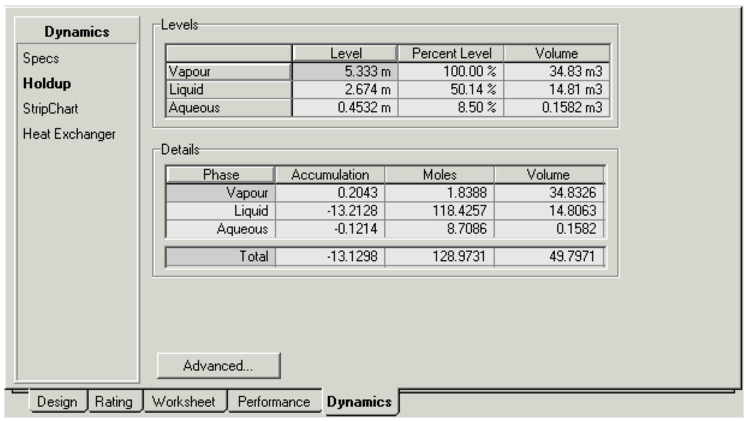
- Kp = 1.8
- Ti = 4 * Residence time / Kp

Holdup Page

Refer to **Section 1.3.3 - Holdup Page** for more information.

The Holdup page contains information regarding the properties, composition, and amount of the holdup.

Figure 2.110



The Levels group displays the following variables for each of the phases available in the vessel:

- Level. Height location of the phase in the vessel.
- Percent Level. Percentage value location of the phase in the vessel.
- Volume. Amount of space occupied by the phase in the vessel.

StripChart Page

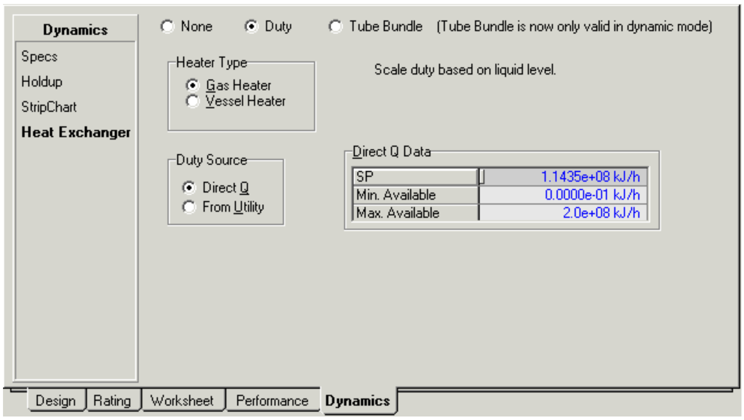
Refer to **Section 1.3.7 - Stripchart Page/Tab** for more information.

The Stripchart page allows you to select and create default strip charts containing various variable associated to the operation.

Heat Exchanger Page

The Heat Exchanger page opens a list of available heating methods for the unit operation. This page contains different objects depending on which configuration you select.

Figure 2.111



Refer to **Duty Radio Button** for more information.

Refer to **Tube Bundle Radio Button** for more information.

- If you select the **None** radio button, this page is blank and the Condenser has no cooling source.
- If you select the **Duty** radio button, this page contains the standard cooling parameters and you have to specify an energy stream for the Condenser.
- If you select the **Tube Bundle** radio button, this page contains the parameters used to configure a kettle chiller and you have to specify the required material streams for the kettle chiller.

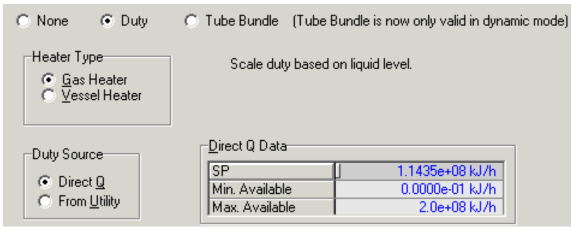
The Tube Bundle options are only available in Dynamics mode.

If you switch from Duty option or Tube Bundle option to None option, HYSYS automatically disconnects the energy or material streams associated to the Duty or Tube Bundle options.

Duty Radio Button

When the Duty radio button is selected the following heat transfer options are available.

Figure 2.112



The Heater Type group has two radio buttons:

- **Gas Heater**. When you select this radio button, the duty is linearly reduced so that it is zero at liquid percent level of 100%, unchanged at liquid percent level of 50%, and doubled at liquid percent level of 0%.
  The following equation is used:

$$Q = (2 - 0.02L)Q_{Total} \tag{2.11}$$

where:

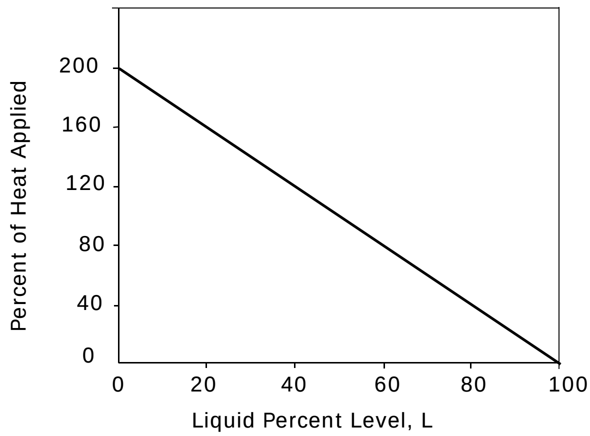
Q = total heat applied to the holdup

L = liquid percent level

Q_{Total} = *duty calculated from the duty source*

The heat applied to the Condenser operation directly varies with the surface area of vapour contacting the vessel wall.

Figure 2.113



Percent Heat Applied to Condenser

The Gas Heater method is available only for condensers, because the heat transfer in the Condenser depends more on the surface area of the vapour contacting the cooling coils than the liquid.

- **Vessel Heater**. When you select this radio button, 100% of the duty specified or calculated in the **SP** cell is applied to the vessel's holdup. That is:

$$Q = Q_{Total} \tag{2.12}$$

where:

Q = *total heat applied to the holdup*

Q_{Total} = *duty calculated from the duty source*

The Vessel Heater method is a non-scaling method.

The Duty Source group has two radio buttons:

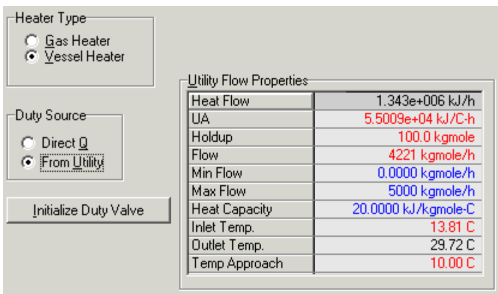
- Direct Q
- From Utility

When you select the Direct Q radio button, the Direct Q Data group appears. The following table describes the purpose of each object in the group.

Object	Description
SP	The heat flow value in this cell is the same value specified in the Duty field on the Parameters page of the Design tab. Any changes made in this cell are reflected on the Duty field on the Parameters page of the Design tab.
Min. Available	Allows you to specify the minimum amount of heat flow.
Max. Available	Allows you to specify the maximum amount of heat flow.

When you select the From Utility radio button, the Utility Flow Properties group appears.

Figure 2.114



The cells containing:

- black text indicates the value is calculated by HYSYS and cannot be changed.
- blue text indicates the value is entered by you, and you can change the value.
- red text indicates the value is calculated by HYSYS, and you can change the value.

The following table describes the purpose of each object that appears when the From Utility radio button is selected.

Object	Description
Heat Flow	Displays the heat flow value.
UA	Displays the overall heat transfer coefficient.
Holdup	Displays the amount of holdup fluid in the condenser.
Flow	Displays the amount of fluid flowing out of the condenser.
Min. Flow	Displays the minimum amount of fluid flowing out of the condenser.
Max. Flow	Displays the maximum amount of fluid flowing out of the condenser.
Heat Capacity	Displays the heat capacity of the fluid.

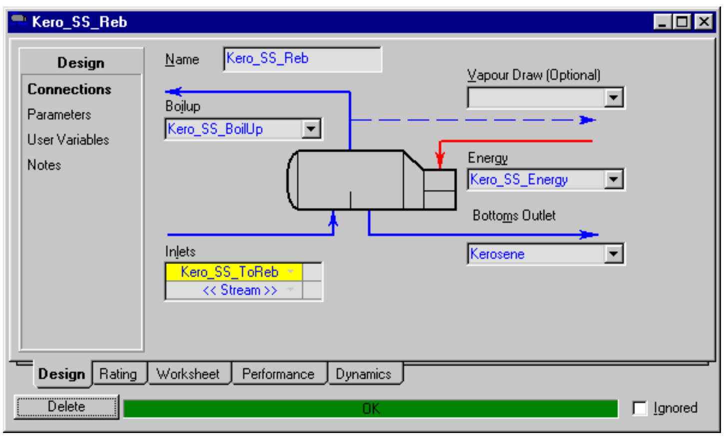
Object	Description
Inlet Temp.	Displays the temperature of the stream flowing into the condenser.
Outlet Temp.	Displays the temperature of the stream flowing out of the condenser.
Temp Approach	Displays the value of the operation outlet temperature minus the outlet temperature of the Utility Fluid. It is only used when one initializes the duty valve via the Initialize Duty Valve button.
Initialize Duty Valve	Allows you to initialize the UA, flow, and outlet temperature to be consistent with the duty for purposes of control.

2.7.2 Reboiler

If you choose a Reboiled Absorber or Distillation template, it includes a Reboiler which is connected to the bottom tray in the tray section with the streams to reboiler and boilup.

The Reboiler is a column operation, where the liquid from the bottom tray of the column is the feed to the reboiler, and the boilup from the reboiler is returned to the bottom tray of the column.

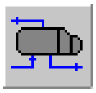
Figure 2.115



The Reboiler is used to partially or completely vapourize liquid feed streams. You must be in a Column subflowsheet to install the Reboiler.

To install the Reboiler operation do one of the following:

- In the Column environment, press **F12** and select Reboiler from the UnitOps property view.
- In the Column environment, press **F4** and click the Reboiler icon in the Column Palette.



Reboiler icon

The Reboiler property view has the same basic tabs that are available on any unit operation:

- Design
- Rating
- Worksheet
- Performance
- Dynamics

It is necessary to specify the connections, and the parameters for the Reboiler. The information on the Dynamics tab are not relevant in steady state.

Design Tab

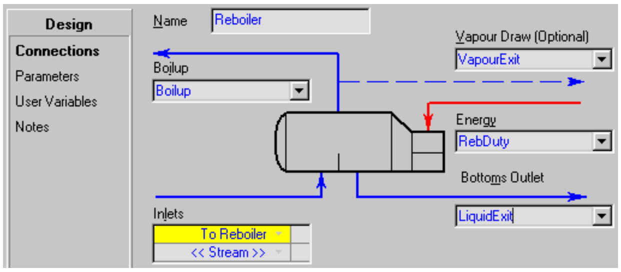
The Design tab contains the following pages:

- Connections
- Parameters
- User Variables
- Notes

Connections Page

On the Connections page, you must specify the Reboiler name, as well as the feed(s), boilup, vapor draw, energy, and bottoms product streams. The vapor draw stream is optional.

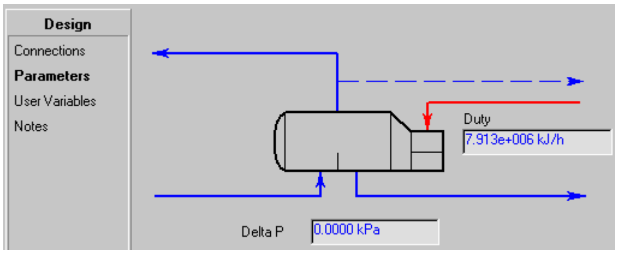
Figure 2.116



Parameters Page

On the Parameter page, you can specify the pressure drop and energy used by the Reboiler. The pressure drop across the Reboiler is zero by default.

Figure 2.117



The Duty for the energy Stream should be positive, indicating that energy is being added to the Reboiler feed(s). If you specify the duty, a degree of freedom is used.

It is recommended to define a duty specification on the Monitor page or Specs page of the Column property view, instead of specifying a value for the duty stream.

The steady state reboiler energy balance is defined as:

$$H_{feed} + Duty = H_{vapour} + H_{bottom} + H_{boilup} \tag{2.13}$$

where:

H_{feed} = *heat flow of the feed stream to the reboiler*

H_{vapour} = *heat flow of the vapour draw stream*

$H_{bottoms}$ = *heat flow of the bottoms product stream*

H_{boilup} = *heat flow of the boilup stream*

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

Rating Tab

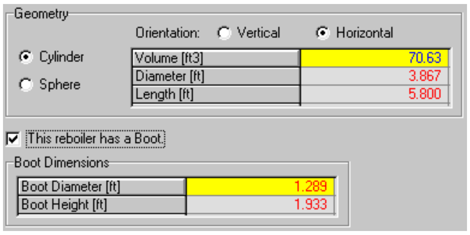
The Rating tab contains the following pages:

- Sizing
- Nozzles
- Heat Loss

Rating tab for a Reboiler is the same as the Rating tab for the Condenser.

Sizing Page

Figure 2.118



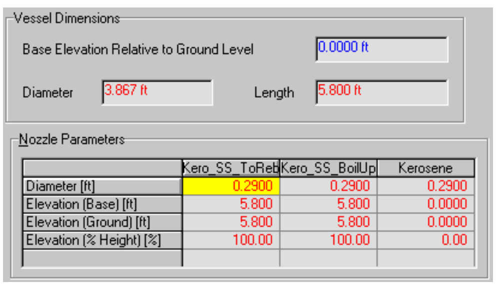
The Sizing page contains all the required information for correctly sizing the reboiler. You can select either vertical or horizontal orientation, and cylinder or sphere. You can either enter the volume or dimensions for your reboiler. You can also indicate whether or not the reboiler has a boot associated with it. If it does, you can specify the boot dimensions.

Nozzles Page

Refer to **Section 1.3.6 - Nozzles Page** for more information.

The Nozzles page contains information regarding the elevation and diameter of the nozzles. The information provided in the Nozzles page is applicable only in Dynamic mode.

Figure 2.119



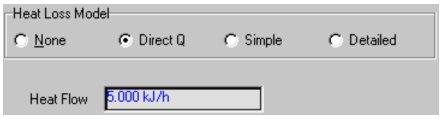
Heat Loss Page

The Heat Loss page allows you to specify the heat loss from individual trays in the tray section. You can choose either a Direct Q, Simple or Detailed heat loss model or no heat loss from the Heat Loss Mode group.

Direct Q Heat Loss Model

The Direct Q model allows you to either specify the heat loss directly, or have the heat loss calculated from the Heat Flow for the reboiler.

Figure 2.120

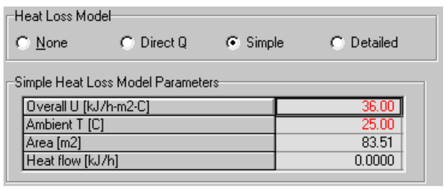


Simple Heat Loss Model

The Simple model allows you to calculate the heat loss from these specified values:

- Overall U value
- Ambient Temperature

Figure 2.121

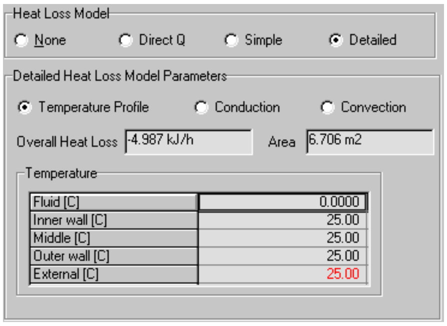


Detailed Heat Loss Model

Refer to **Section 1.6.1 - Detailed Heat Model** in the **HYSYS Dynamic Modeling** guide for more information.

The Detailed model allows you to specify more detailed heat transfer parameters.

Figure 2.122



Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the Reboiler.

The PF Specs page is relevant to dynamics cases only.

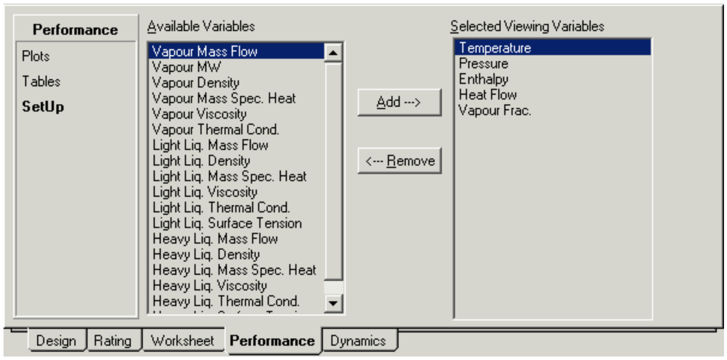
Performance Tab

The Performance tab of the Reboiler has the same pages as the Performance tab of the Condenser:

- Plots
- Tables

- SetUp

Figure 2.123



From these pages you can select the type of variables you want to calculate and plot, view the calculated values, and plot any combination of the selected variables. The default selected variables are temperature, pressure, heat flow, enthalpy, and vapor fraction. At the bottom of the Plots or Tables page, you can specify the interval size over which the values should be calculated and plotted.

Dynamics Tab

The Dynamics tab contains the following pages:

- Specs
- Holdup
- StripChart
- Heat Exchanger

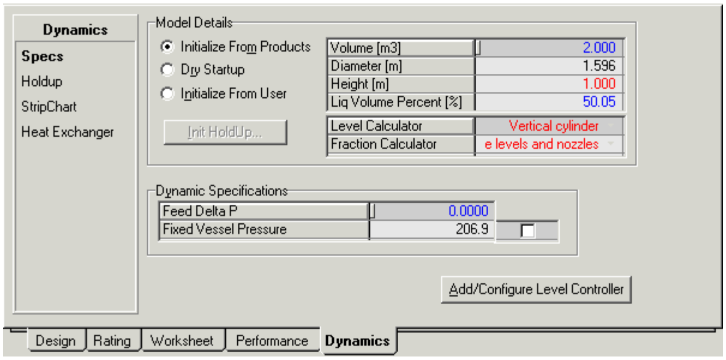
The Dynamics tab for a Reboiler is the same as the Dynamics tab for the Condenser.

You are not required to modify information on the Reboiler's Dynamics tab when working in Steady State mode.

Specs Page

The Specs page contains information regarding initialization modes, reboiler geometry, and reboiler dynamic specifications.

Figure 2.124



Model Details

In the Model Details group, you can specify the initial composition and amount of liquid that the separator should start with when you start dynamics. This done via the initialization mode which is discussed in the table below.

Initialization Mode	Description
Initialize from Products	The composition of the holdup is calculated from a weighted average of all products exiting the holdup. A PT flash is performed to determine other holdup conditions. The liquid level is set to the value indicated in the Liq Volume Percent field.
Dry Startup	The composition of the holdup is calculated from a weighted average of all feeds entering the holdup. A PT flash is performed to determine other holdup conditions. The liquid level in the Liq Volume Percent field is set to zero.
Initialize from User	The composition of the liquid holdup in the reboiler is user specified. The molar composition of the liquid holdup can be specified by clicking the Init Holdup button. The liquid level is set to the value indicated in the Liq Volume Percent field.

The Initialization Mode can be changed any time when the integrator is not running. The changes cause the vessel to re-initialize when the integrator is started again.

The reboiler geometry can be specified in the Model Details group. The following reboiler geometry parameters can be specified in the same manner as the Geometry group on the Sizing page of the Rating tab:

- Volume
- Diameter
- Height (Length)
- Geometry (Level Calculator)

The Liquid Volume Percent value is also displayed in this group. You can modify the level in the condenser at any time. HYSYS then uses that level as an initial value when the integrator is run.

Refer to the section on the **Nozzles Page** for more information.

The Fraction Calculator determines how the level in the condenser, and the elevation and diameter of the nozzle affects the product composition. There is only one Fraction Calculation mode available, it is called Use Levels and Nozzles. The calculations are based on how the nozzle location and vessel liquid level affect the product composition.

Dynamic Specifications

The Dynamic Specifications group contains fields where you can specify what happens to the pressure of the reboiler when you enter dynamic mode.

The Feed Delta P field allows you to impose a fixed pressure drop between the vessel and all of the feed streams. This is mostly supported for compatibility with Steady State mode. In Dynamic mode, you are advised to properly account for all pressure losses by using the appropriate equipment such as valves or pumps or static head contributions. A zero pressure drop should preferably be used here otherwise you may get unrealistic results such as material flowing from a low to a high pressure area.

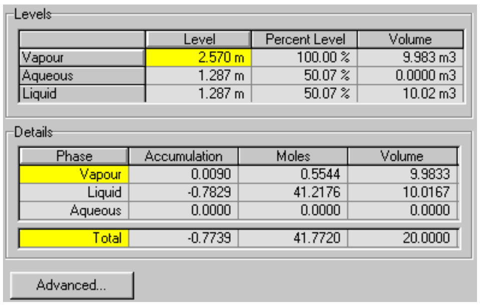
The Fixed Vessel Pressure field allows you to fix the vessel pressure in Dynamic mode. This option can be used in simpler models where you do not want to configure pressure controllers and others, or if the vessel is open to the atmosphere. In general the specification should not be used, because the pressure should be determined by the surrounding equipment.

Holdup Page

Refer to **Section 1.3.3 - Holdup Page** for more information.

The Holdup page contains information regarding the properties, composition, and amount of the holdup.

Figure 2.125



The Levels group displays the following variables for each of the phases available in the vessel:

- Level. Height location of the phase in the vessel.
- Percent Level. Percentage value location of the phase in the vessel.
- Volume. Amount of space occupied by the phase in the vessel.

StripChart Page

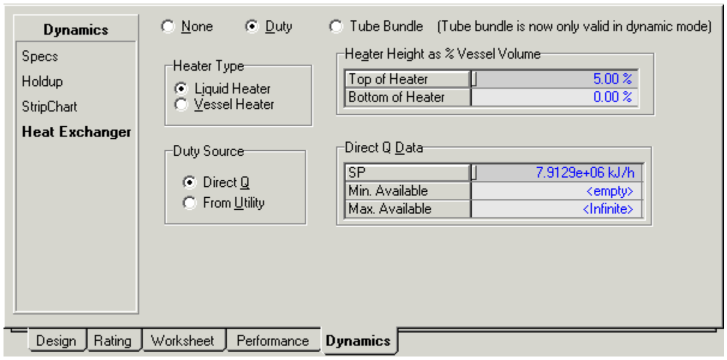
Refer to **Section 1.3.7 - Stripchart Page / Tab** for more information.

The Stripchart page allows you to select and create default strip charts containing various variable associated to the operation.

Heat Exchanger Page

The Heat Exchanger page opens a list of available heating methods for the unit operation. This page contains different objects depending on which radio button you select.

Figure 2.126



Refer to **Duty Radio Button** for more information.

Refer to **Tube Bundle Radio Button** for more information.

- If you select the **None** radio button, this page is blank and the Condenser has no cooling source.
- If you select the **Duty** radio button, this page contains the standard heating parameters and you have to specify an energy stream for the Reboiler.
- If you select the **Tube Bundle** radio button, this page contains the parameters used to configure a kettle reboiler and you have to specify the required material streams for the kettle reboiler.

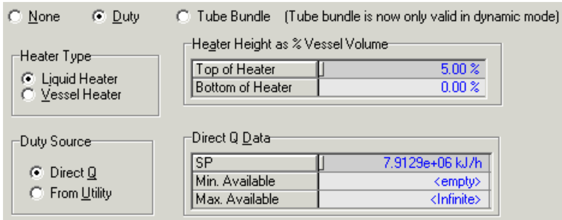
The Tube Bundle options are only available in Dynamics mode.

If you switch from Duty option or Tube Bundle option to None option, HYSYS automatically disconnects the energy or material streams associated to the Duty or Tube Bundle options.

Duty Radio Button

When the Duty radio button is selected the following heat transfer options are available.

Figure 2.127



The Heater Type group has two radio buttons:

- Liquid Heater

When you select the Liquid Heater radio button, the Heater Height as % Vessel Volume group appears. This group contains two cells:

- **Top of Heater**
- **Bottom of Heater**

These cells are used to specify the heater height.

- Vessel Heater

For the Liquid Heater method, the duty applied to the vessel depends on the liquid level in the tank. The heater height value must be specified. The heater height is expressed as a percentage of the liquid level in the vessel operation. The default values are 5% for the top of the heater, and 0% for the bottom of the heater. These values are used to scale the amount of duty that is applied to the vessel contents.

$$\begin{aligned} Q &= 0 && (L < B) \\ Q &= \frac{L-B}{T-B} Q_{Total} && (B \le L \le T) \\ Q &= Q_{Total} && (L > T) \end{aligned} \qquad (2.14)$$

where:

L = liquid percent level (%)

T = top of heater (%)

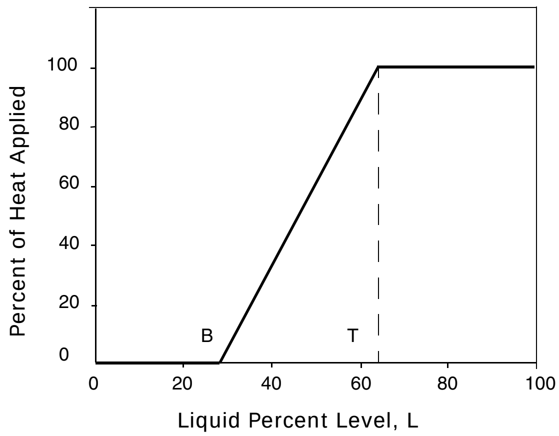
B = bottom of heater (%)

The Percent Heat Applied may be calculated as follows:

$$Percent\ Heat\ Applied = \frac{Q}{Q_{Total}} \times 100\% \tag{2.15}$$

It is shown that the percent of heat applied to the vessel's holdup directly varies with the surface area of liquid contacting the heater.

Figure 2.128



Percent Heat Applied for a Liquid Heater

When you select the Vessel Heater radio button, 100% of the duty specified or calculated in the SP cell is applied to the vessel's holdup:

$$Q = Q_{Total} \tag{2.16}$$

where:

Q = *total heat applied to the holdup*

Q_{Total} = *duty calculated from the duty source*

The Duty Source group has two radio buttons:

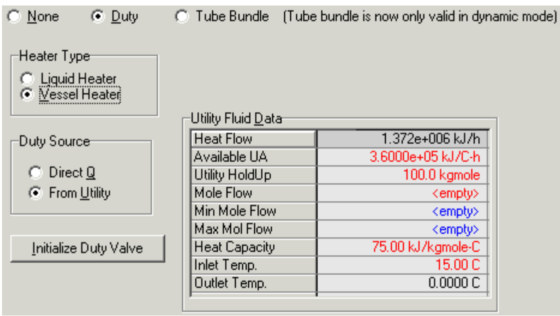
- Direct Q
- From Utility

When you select the Direct Q radio button, the Direct Q Data group appears. The following table describes the purpose of each object in the group.

Object	Description
SP	The heat flow value in this cell is the same value specified in the Duty field of the Parameters page on the Design tab. Any changes made in this cell is reflected on the Duty field of the Parameters page on the Design tab.
Min. Available	Allows you to specify the minimum amount of heat flow.
Max. Available	Allows you to specify the maximum amount of heat flow.

When you select the From Utility radio button, the Utility Flow Properties group appears.

Figure 2.129



The cells containing:

- black text indicates the value is calculated by HYSYS and cannot be changed.
- blue text indicates the value is entered by you, and you can change the value.
- red text indicates the value is calculated by HYSYS, and you can change the value.

The following table describes the purpose of each object that appears when the From Utility radio button is selected.

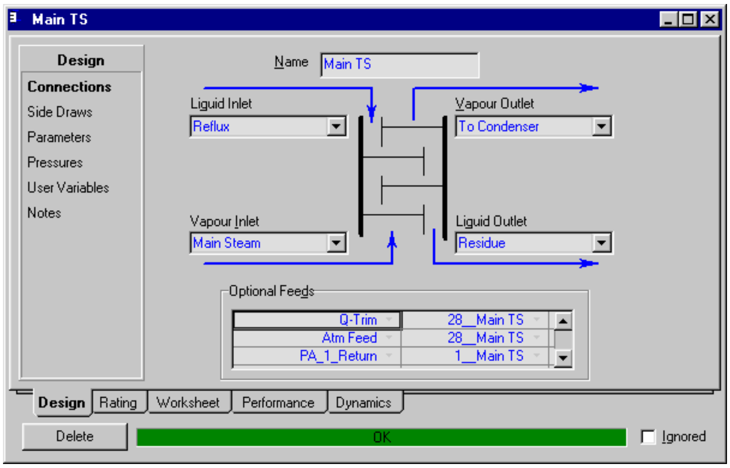
Object	Description
Heat Flow	Displays the heat flow value.
Available UA	Displays the overall heat transfer coefficient.
Utility Holdup	Displays the amount of holdup fluid in the reboiler.
Mole Flow	Displays the amount of fluid flowing out of the reboiler.
Min Mole Flow	Displays the minimum amount of fluid flowing out of the reboiler.
Max Mole Flow	Displays the maximum amount of fluid flowing out of the reboiler.
Heat Capacity	Displays the heat capacity of the fluid.

Object	Description
Inlet Temp.	Displays the temperature of the stream flowing into the condenser.
Outlet Temp.	Displays the temperature of the stream flowing out of the condenser.
Initialize Duty Valve	Allows you to initialize the UA, flow, and outlet temperature to be consistent with the duty for purposes of control.

2.7.3 Tray Section

At the very minimum, every Column Templates includes a tray section. An individual tray has a vapour feed from the tray below, a liquid feed from the tray above, and any additional feed, draw or duty streams to or from that particular tray. The property view for the tray section of a Distillation Column template is shown in the figure below.

Figure 2.130



The tray section property view contains the five tabs that are common to most unit operations:

- Design
- Rating
- Worksheet
- Performance
- Dynamics

You are not required to change anything on the Rating tab and Dynamics tab, if you are operating in Steady State mode.

Design Tab

The Design tab contains the following pages:

- Connections
- Side Draws
- Parameters
- Pressures
- User Variables
- Notes

Connections Page

The Connections page of the Tray Section is used for specifying the names and locations of vapour and liquid inlet and outlet streams, feed streams, and the number of stages (see **Figure 2.130**). When a Column template is selected, HYSYS inserts the default stream names associated with the template into the appropriate input cells. For example, in a Distillation Column, the Tray Section vapour outlet stream is To Condenser and the Liquid inlet stream is Reflux.

A number of conventions exist for the naming and locating of streams associated with a Column Tray Section:

- When you select a Tray Section feed stream, HYSYS by default feeds the stream to the middle tray of the column (for example, in a 20-tray column, the feed would enter on tray 10). The location can be changed by selecting the desired feed tray from the drop-down list, or by typing the tray number in the appropriate field.
- Streams entering and leaving the top and bottom trays are always placed in the Liquid or Vapor Inlet/Outlet fields.

Specifying the location of a column feed stream to be either the top tray (tray 1 or tray N, depending on your selected numbering convention) or the bottom tray (N or 1) automatically results in the stream becoming the Liquid Inlet or the Vapour Inlet, respectively. If the Liquid Inlet or Vapour Inlet already exists, your specified feed stream is an additional stream entering on the top or bottom tray, displayed with the tray number (1 or N). A similar convention exists for the top and bottom tray outlet streams (Vapour Outlet and Liquid Outlet).

Side Draws Page

On the Side Draws page, you can specify the name and type of side draws taken from the tray section of your column. Use the radio buttons to select the type of side draw:

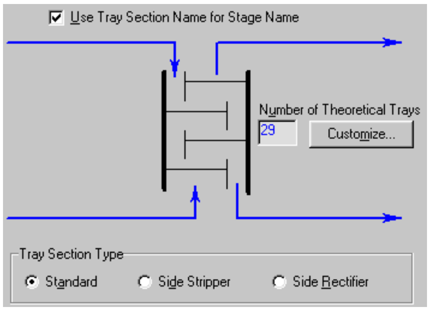
- Vapor
- Liquid
- Water

Select the cells to name the side draw stream, and specify the tray from which it is taken.

Parameters Page

You can input the number of trays on the Parameters page.

Figure 2.131

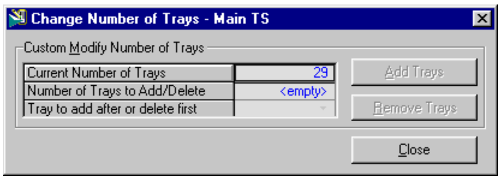


By default, the Use Tray Section Name for Stage Name checkbox is selected.

The trays are treated as ideal if the fractional efficiencies are set to 1. If the efficiency of a particular tray is less than 1, the tray is modeled using a modified Murphree Efficiency.

You can add or delete trays anywhere in the column by clicking the Customize button, and entering the appropriate information in the Custom Modify Number of Trays group. This feature makes adding and removing trays simple, especially if you have a complex column, and you do not want to lose any feed or product stream information. The figure below shows the property view that appears when the Customize button is clicked.

Figure 2.132



You can add and remove trays by:

- Specify a new number of trays in the **Current Number of Trays** field.

  This is the same as changing the number of theoretical trays on the **Connections** page. All inlet and outlet streams move appropriately; for example, if you are changing the number of trays from 10 to 20, a stream initially connected to tray 5 is now at tray 10, and a stream initially connected at stream 10 is now at tray 20.
- Add or remove trays into or from individual tray section.

When you are adding or deleting trays, all Feeds remain connected to their current trays.

Adding Trays

To add trays to the tray section:

1. Enter the number of trays you want to add in the Number of Trays to Add/Delete field.
2. Specify the tray number after, which you want to add the trays in the Tray to Add After or Delete First field.
3. Click the **Add Trays** button, and HYSYS inserts the trays in the appropriate place according to the tray numbering sequence you are using. All streams (except feeds) and auxiliary equipment below (or above, depending on the tray numbering scheme) the tray where you inserted is moved down (or up) by the number of trays that were inserted.

Removing Trays

To remove trays from the tray section:

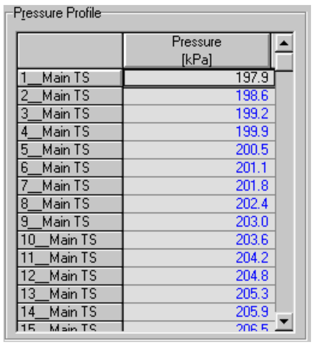
1. Enter the number of trays you want to delete in the Number of Trays to Add/Delete field.
2. Enter the first tray in the section you want to delete in the Tray to Add After or Delete First field.
3. Click the **Remove Trays** button. All trays in the selected section are deleted. If you are using the top-down numbering scheme, the appropriate number of trays *below* the first tray (and including the first tray) you specify are removed. If you are using the bottom-up scheme, the appropriate number of trays *above* the first tray (and including the first tray) you specify are removed.
4. Streams connected to a higher tray (numerically) are not affected; for example, if you are deleting 3 trays starting at tray number 6, a side draw initially at tray 5 remains there, but a side draw initially connected to tray 10 is now at tray 7. Any draw streams connected to trays 6,7 or 8 are deleted with your confirmation to do so.

If you select the Side Stripper radio button or Side Rectifier radio button at the bottom of the property view, this affects the pressure profile. The pressure of the main tray section stage from which the liquid feed stream is drawn is used as the side stripper pressure, which is constant for all stages. The pressure of the main tray section stage from which the vapour feed stream is drawn is used as the Side Rectifier pressure, which is constant for all stages.

Pressures Page

The Pressures page displays the pressure on each tray. Whenever two pressures are known for the tray section, HYSYS interpolates to find the intermediate pressures. For example, if you enter the Condenser and Reboiler Pressures through the Column Input Expert or Column property view, HYSYS calculates the top and bottom tray pressures based on the Condenser and Reboiler pressure drops. The intermediate tray pressures are then calculated by linear interpolation.

Figure 2.133



Pressure Profile

	Pressure [kPa]
1__Main TS	197.9
2__Main TS	198.6
3__Main TS	199.2
4__Main TS	199.9
5__Main TS	200.5
6__Main TS	201.1
7__Main TS	201.8
8__Main TS	202.4
9__Main TS	203.0
10__Main TS	203.6
11__Main TS	204.2
12__Main TS	204.8
13__Main TS	205.3
14__Main TS	205.9
15__Main TS	206.5

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

Rating Tab

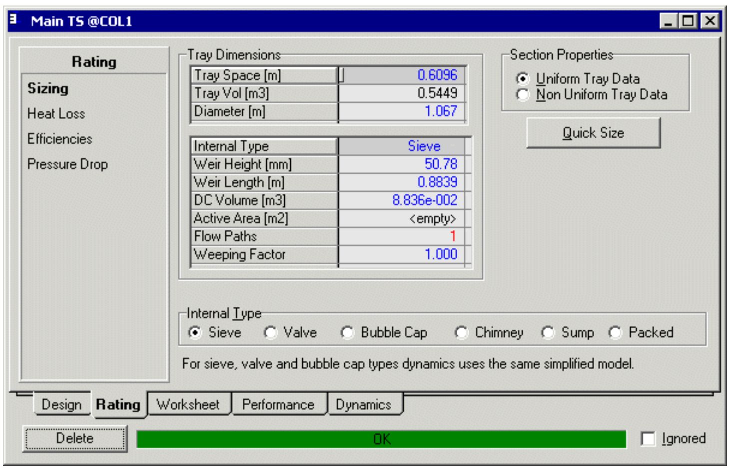
The Rating tab contains the following pages:

- Sizing
- Nozzles
- Heat Loss
- Efficiencies
- Pressure Drop

Sizing Page

The Sizing page contains the required information for correctly sizing column tray and packed sections. If the Sieve, Valve, Bubble Cap radio button with the Uniform Tray Data are selected, the following property view is shown.

Figure 2.134



Each parameter is also discussed in **Section 14.19 - Tray Sizing**.

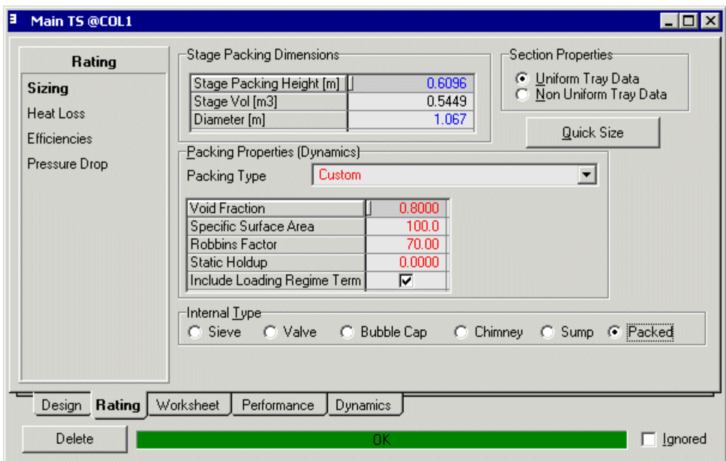
The tray section diameter, weir length, weir height, and the tray spacing are required for an accurate and stable dynamic simulation. You must specify all of the information on this page. The Quick Size button allows you to automatically and quickly size the tray parameters. The Quick Size calculations are based on the same calculations that are used in the Tray Sizing Utility.

The required size information for the tray section can be calculated using the Tray Sizing utility.

HYSYS only calculates the tray volume, based on the weir length, tray spacing, and tray diameter. For multipass trays, simply enter the column diameter and the appropriate total weir length.

When you select the Packed radio button and the Uniform Tray Data section, the Sizing page changes to the property view shown below.

Figure 2.135



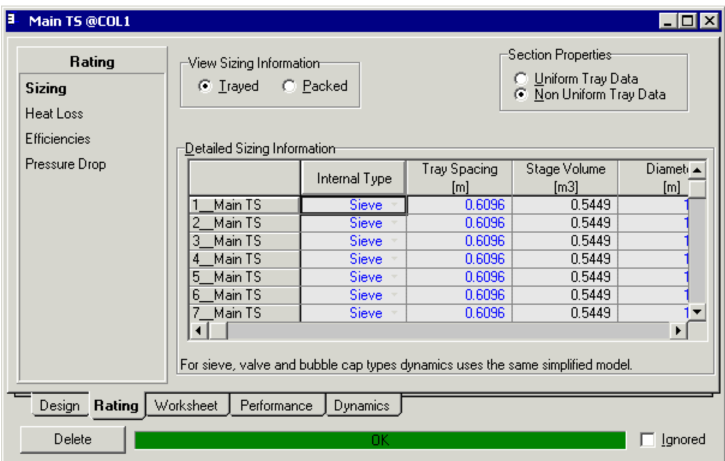
The stage packing height, stage diameter, packing type, void fraction, specified surface area, and Robbins factor are required for the simple dynamic model. HYSYS uses the stage packing dimensions and packing properties to calculate the pressure flow relationship across the packed section.

Packing Properties (Dynamics)	Description
Void Fraction	Packing porosity, in other words, m^3 void space/ m^3 packed bed.
Specific Surface Area	Packing surface area per unit volume of packing (m^{-1}).

Packing Properties (Dynamics)	Description
Robbins Factor	A packing-specific quantity used in the Robbins correlation, which is also called the dry bed packing factor (m^{-1}). The Robbins correlation is used to predict the column vapour pressure drop. For the dry packed bed at atmospheric pressure, [2] the Robbins or packing factor is proportional to the vapour pressure drop.
Static Holdup	Static liquid, h_{st}, is the m^3 liquid/ m^3 packed bed remaining on the packing after it has been fully wetted and left to drain. The static liquid holdup is a constant value.
Include Loading Regime Term	Loading regime term is the second term in the Robbins pressure drop equation, which is limited to atmospheric pressure and under vacuum but not at elevated pressures. When pressure is high, (in other words, above 1 atm), inclusion of the loading regime term may cause an unrealistically high pressure drop prediction.

To specify Chimney and Sump tray types, the Non Uniform Tray Data Option must be selected from the Section Properties group. The Non Uniform Tray Data Option allows you to model a column with high fidelity by adjusting tray rating parameters on a tray by tray basis.

Figure 2.136



For a Trayed section of a column, you can adjust the Internal Type of tray, Tray Spacing, Diameter, Weir Height, Weir Length, DC Volume, Flow Path and Weeping factor. For a Packed section of a column, you can adjust the Stage Packing Height, and Diameter.

From the Internal Type drop-down list in the Detailed Sizing Information group, you can select alternative internal tray types on a tray by tray basis.

The Chimney and Sump internals along with the weeping factor details are mentioned below.

Detailed Sizing Information	Description
Internal Type	**Chimney -** This allows a higher liquid level and does not have any liquid going down to the tray below. Although vapor can go up through it but it does not contact the liquid. The Chimney tray type can be designated on any tray. By default, the weeping factor is set to 0 and the stage efficiency is set to 5% on the Efficiencies page. The weir height and tray spacing is increased for a tray section. For a packed section stage packing height is increased. **Sump** - Only the bottom tray can be designated as a sump. By default, the efficiency is set to 5%. The tray spacing for a tray section and the stage packing height in a packed section are increased when using a Sump.
Weeping Factor	The weeping factor can be adjusted on a tray by tray basis. It is used to scale back or turn off weeping. By default the weeping factor is set to 1 for all internal types except the sump.

Nozzles Page

Refer to **Section 1.3.6 - Nozzles Page** for more information.

The Nozzles page contains the elevations at which vapour and liquid enter or leave the tray section.

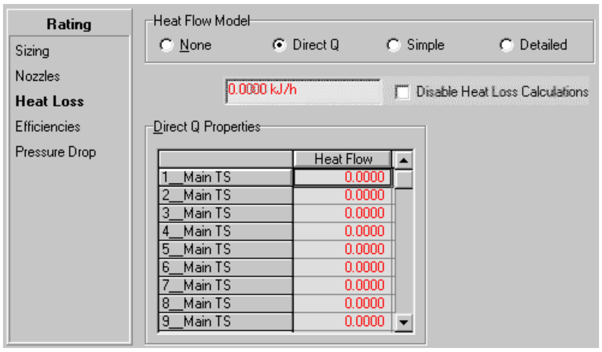
Heat Loss Page

The Heat Loss page allows you to specify the heat loss from individual trays in the tray section. You can select from either a Direct Q, Simple or Detailed heat loss model or have no heat loss from the tray sections.

Direct Q Heat Flow Model

The Direct Q model allows you to input the heat loss directly where the heat flow is distributed evenly over each tray section. Otherwise you have the heat loss calculated from the Heat Flow for each specified tray section.

Figure 2.137



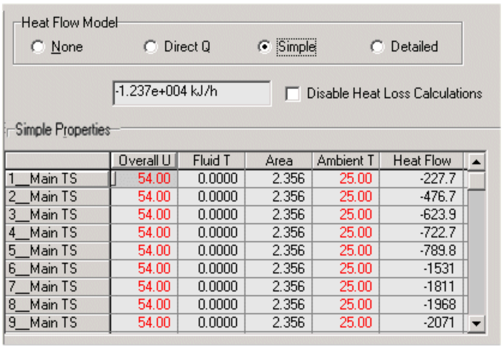
Using the checkbox, you can temporarily disable heat loss calculations without losing any Heat Loss data that is entered.

Simple Heat Flow Model

The Simple model allows you to calculate the heat loss by specifying:

- The Overall U value
- The Ambient Temperature°C

Figure 2.138

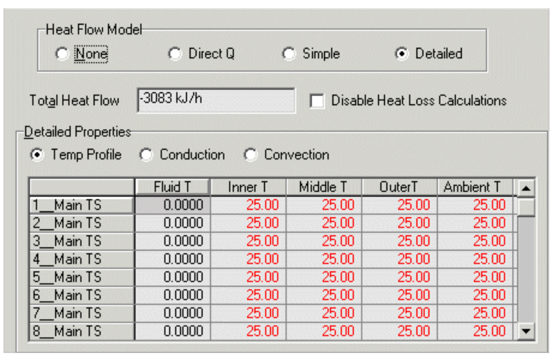


Detailed Heat Flow Model

Refer to **Section 1.6.1 - Detailed Heat Model** in the **HYSYS Dynamic Modeling** guide for more information.

The Detailed Heat Flow model allows you to specify more detailed heat transfer parameters. The detailed properties can be used on a tray to tray basis based on the temperature profile, conduction, and convection data specified.

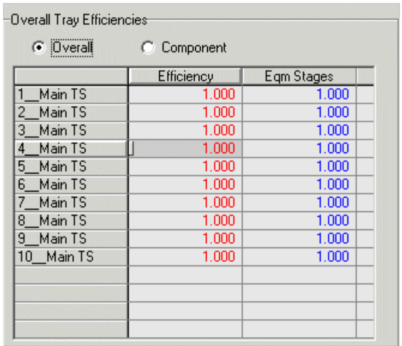
Figure 2.139



Efficiencies Page

As with steady state, you can specify tray efficiencies for columns in dynamics. However, you can only specify the overall tray efficiency; component tray efficiencies are only available in steady state.

Figure 2.140



Pressure Drop Page

The Pressure Drop page displays the information associated with the pressure drops (or pressures) across the tray section.

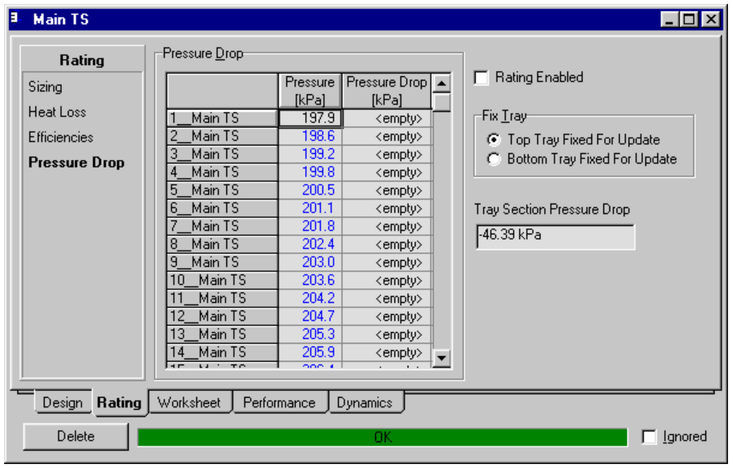
The Pressure Drop page uses the same calculation in the Tray Sizing utility to calculate the pressure drop for the tray sections when the column is running. In other words, using the traffics and geometries to determine what the pressure drop is.

Selecting the **Rating Enabled** checkbox turns on the pressure drop calculations as part of the column solution.

The tray sizing utility calculates a pressure drop across each tray, you need to fix one end of the column (top or bottom), allowing the other trays to float with the calculations. You can select which end of the column to be fixed by selecting the appropriate radio button in the Fix Tray group.

The Tray Section Pressure Drop field displays the absolute overall pressure change between the fixed tray and the last tray at the other end.

Figure 2.141



Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the Tray Section.

The PF Specs page is relevant to dynamics cases only.

Performance Tab

The Performance tab contains the following pages:

- Pressure
- Temperature
- Flow
- Summary
- Hydraulics

Pressure Page

The Pressure page contains a table that lists all the pressure for each tray. The table also includes the names of any inlet streams associated to a tray and the inlet streams’ pressure.

Temperature Page

The Temperature page contains a table that lists all the temperature for each tray. The table also includes the names of any inlet streams associated to a tray and the inlet streams’ temperature.

Flow Page

The Flow page contains a table that lists all the liquid and vapour flow rates for each tray. The table also includes the names of any inlet streams associated to a tray and the inlet streams’ flow rate. You can also change the unit of the flow rates displayed by selecting the unit from the Flow Basis drop-down list. There are four possible units:

- Molar
- Mass
- Standard Liquid Volume
- Actual Volume

Summary Page

The Summary page contains a table that displays the flow rates, temperature, and pressure for each tray.

Hydraulics Page

The Hydraulics page contains a table that displays the height and pressure of Dry Hole DP, Static Head, and Height over Weir.

The Hydraulics page is only available in dynamic mode.

Dynamics Tab

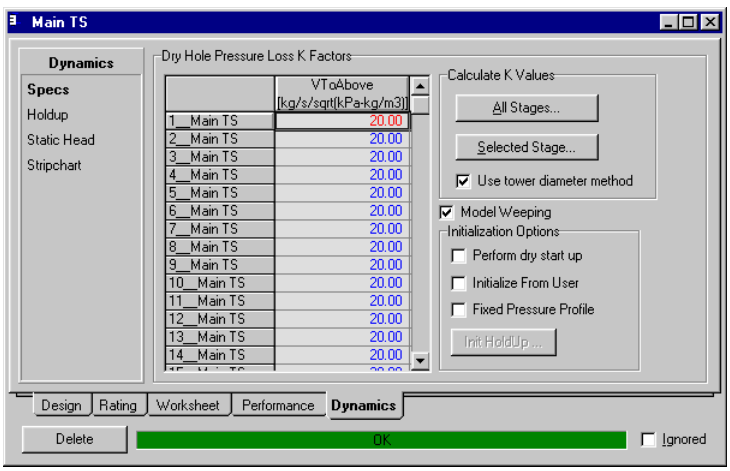
The Dynamics tab contains the following pages:

- Specs
- Holdup
- Static Head
- StripChart

Specs Page

The Specs page contains the *Nozzle Pressure Flow k Factors* for all the trays in the tray section. You can select to have HYSYS calculate the k value for all the trays by clicking the All Stages button. If you want HYSYS to calculate the k values for certain trays only, select the desired trays and click the Selected Stages button. HYSYS only calculates the k values for the selected stages.

Figure 2.142



The **Use tower diameter method** checkbox, when selected, calculates the k values for the column based on the column diameter. When the checkbox is cleared the k values are calculated using the results obtained from the steady state model, providing a smoother transition between your steady state model and dynamic model.

The **Model Weeping** checkbox, when selected, takes into account any weeping that occurs on the tray sections and add the effects to your model.

Weeping can start to occur on a tray when the dry hole pressure loss drops below 0.015 kPa. It allows liquid to drain to the stage below even if the liquid height is below the weir height.

The **Perform dry start up** checkbox allows you to simulate a dry start up. Selecting this checkbox removes all the liquid from all the trays when the integrator starts.

The **Initialize From User** checkbox allows you to start the simulation from conditions you specify. Selecting this checkbox, activates the **Init HoldUp** button. Click this button to enter the initial liquid mole fractions of each component and the initial flash conditions.

The Fixed Pressure Profile checkbox allows you to simulate the column based on the fixed pressure profile.

Pressure Profile

The Fixed Pressure Profile checkbox allows you to run the column in Dynamic mode using the steady state pressure profile. This option simplifies the column solution for inexperienced users, and makes their transition from the steady state to dynamics simulation a bit easier.

You do not have to configure pressure control systems with this option. This option is not recommended for rigorous modeling work where the pressure can typically change on response to other events.

The pressure profile of a tray section is determined by the static head, which is caused mostly by the liquid on the trays, and the frictional pressure losses, which are also known as dry hole pressure loses.

The frictional pressure losses are associated with vapour flowing through the tray section. The flowrate is determined by **Equation (2.17)**.

$$\text{flow} = k \times \sqrt{\text{density} \times \text{friction pressure losses}} \tag{2.17}$$

In HYSYS, the k-value is calculated by assuming:

$$k \alpha (\text{Tray diameter})^2 \tag{2.18}$$

However, if the Fixed Pressure Profile option is selected, then the static head contribution can be subtracted and hence the vapour flow and the frictional pressure loss is known. This allows the k-values to be directly calculated to match steady state results more closely.

Holdup Page

The Holdup page contains a summary of the dynamic simulation results for the column. The holdup pressure, total volume, and bulk liquid volume results on a tray basis are contained in this property view. Double-clicking on a stage name in the Holdup column opens the stage property view.

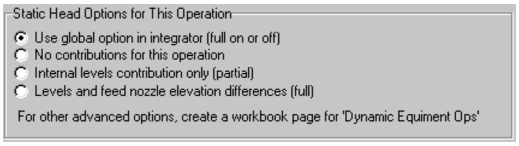
You can double-click on any cell within each row to view the advanced holdup properties for each specific tray section.

Static Head Page

Refer to **Section 1.6.5 - Static Head** in the **HYSYS Dynamic Modeling** guide for more information.

The Static Head page enables you to select how the static head contributes to the calculation.

Figure 2.143



Since static head contributions are often essential for proper column modeling, internal static head contributions are generally considered for the column model in any case, and should only be disabled under special circumstances.

StripChart Page

Refer to **Section 1.3.7 - Stripchart Page / Tab** for more information.

The Stripchart page allows you to select and create default strip charts containing various variable associated to the operation.

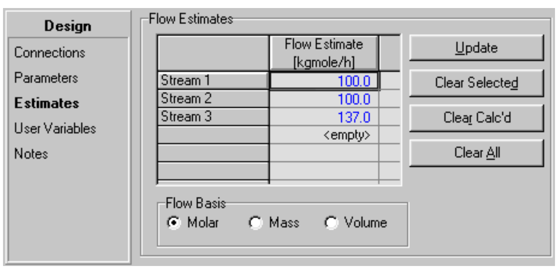
2.7.4 Tee

Refer to **Chapter 7 - Piping Operations** for more details on the property view of the **TEE**.

Refer to the section on the **General Features of the Solving Methods** for information on which method supports Tee operation.

The property view for the Tee operation in the Column subflowsheet has all of the pages and inherent functionality contained by the Tee in the Main Environment with one addition, the Estimates page.

Figure 2.144



On the Estimates page, you can help the convergence of the Column subflowsheet's simultaneous solution by specifying flow estimates for the tee product streams. To specify flow estimates:

1. Select one of the Flow Basis radio buttons: **Molar**, **Mass** or **Volume**.
2. Enter estimates for any of the product streams in the associated fields next to the stream name.

There are four buttons on the Estimates page, which are described in the table below.

Button	Related Setting
Update	Replaces all estimates except user specified estimates (in blue) with values obtained from the solution.
Clear Selected	Deletes the highlighted estimate.
Clear Calculated	Deletes all calculated estimates.
Clear All	Deletes all estimates.

If the Tee operation is attached to the column (for example, via a draw stream), one tee split fraction specification is added to the list of column specifications for each tee product stream that you specify. As you specify the split fractions for the product streams, these values are transferred to the individual column specifications on the Monitor page and Specs page of the column property view.

The additional pieces of equipment available in the Column subflowsheet are identical to those in the main flowsheet. For information on each piece of equipment, refer to its respective chapter.

For example, for information on the Heat Exchanger, refer to Section 4.4 - Heat Exchanger in this manual.

All operations within the Column subflowsheet environment are solved simultaneously.

2.8 Running the Column

Once you are satisfied with the configuration of your Column subflowsheet and you have specified all the necessary input, the next step is to run the Column solution algorithm.

The iterative procedure begins when you click the Run button on the Column property view. The Run/Reset buttons can be accessed from any page of the Column property view.


Run icon


Stop icon

When you are inside the Column build environment, a Run icon also appears on the toolbar, which has the same function as the Run button on the Column property view.

On the toolbar, the Run icon and Stop icon are two separate icons. Whichever icon is toggled on has light grey shading.

When the Run button on the Column property view is clicked, the Run/Reset buttons are replaced by a Stop button which, when clicked, terminates the convergence procedure. The Run button can then be clicked again to continue from the same location. Similarly, the **Stop** icon switches to a grey shading with the **Run** icon on the toolbar after it is activated.

When you are working inside the Column build environment, the Column runs only when you click the Run button on the Column property view, or the Run icon on the toolbar. When you are working with the Column property view in the Main build environment, the Column automatically runs when you change:

- A specification value after a converged solution has been reached.
- The Active specifications, such that the Degrees of Freedom return to zero.

2.8.1 Run

Refer to **Monitor Page** from **Section 2.4.1 - Design Tab** for more information.

The Run command begins the iterative calculations necessary to simulate the column described by the input. On the Monitor page of the Column property view, a summary showing the iteration number, equilibrium error, and the heat and specification errors appear. Detailed messages showing the convergence status are shown in the Trace Window.

The default basis for the calculation is a modified “inside-out” algorithm. In this type of solution, simple equilibrium and enthalpy models are used in the inner loop, which solve the overall component and heat balances, vapour-liquid equilibrium, and any specifications. The outer loop updates the simple thermodynamic models with rigorous calculations.

When the simulation is running, the status line at the bottom of the screen first tracks the calculation of the initial properties used to generate the simple models. Then the determination of a Jacobian matrix appears, which is used in the solution of the inner loop. Next, the status line reports the inner loop errors and the relative size of the step taken on each of the inner loop iterations. Finally, the rigorous thermodynamics is again calculated and the resulting equilibrium, heat, and spec errors reported. The calculation of the inner loop and the outer loop properties continues until convergence is achieved, or you determine that the column cannot converge and click Stop to terminate the calculation.

If difficulty is encountered in converging the inner loop, the program occasionally recalculates the inner loop Jacobian. If no obvious improvement is being made with the printed equilibrium and heat and spec errors, click **Stop** to terminate the calculations and examine the available information for clues.

Refer to **Section 2.9 - Column Troubleshooting** for solutions to some common troubles encountered while trying to achieve the desired solution.

Refer to **Estimates Page** from **Section 2.4.2 - Parameters Tab** for more information.

Any estimates which appear in the Column Profile page and Estimates page are used as initial guesses for the convergence algorithm. If no estimates are present, HYSYS begins the convergence procedure by generating initial estimates.

2.8.2 Reset

The Reset command clears the current Column solution, and any estimates appearing on the Estimates page of the Column property view. If you make major changes after getting a converged Column, it is a good idea to Reset to clear the previous solution. This allows the Column solver to start fresh and distance itself from the previous solution. If you make only minor changes to the Column, try clicking Run before Resetting.

Once the column calculation has started it continues until it has either converged, has been terminated due to a mathematically impossible condition, (for example being unable to invert the Jacobian matrix), or it has reached the maximum number of iterations. Other than these three situations, calculations continue indefinitely in an attempt to solve the column unless the **Stop** button is clicked. Unconverged results can be analysed, as discussed in **Section 2.9 - Column Troubleshooting**.

2.9 Column Troubleshooting

Although HYSYS does not require any initial estimates for convergence, good estimates of top and bottom temperatures and one product accelerate the convergence process. Detailed profiles of vapour and liquid flow rates are not required.

However, should the column have difficulty, the diagnostic output printed during the iterations provides helpful clues on how the tower is performing. If the equilibrium errors are approaching zero, but the heat and spec errors are staying relatively constant, the specifications are likely at fault. If both the equilibrium errors and the heat and spec errors do not appear to be getting anywhere, then examine all your input (for example the initial estimates, the specifications, and the tower configuration).

In running a column, keep in mind that the Basic Column Parameters cannot change. By this, it is meant that column pressure, number of trays, feed tray locations, and extra attachments such as side exchanger and pump around locations remain fixed. To achieve the desired specifications the Column only adjusts variables which have been specified as initial estimates, such as reflux, side exchanger duties, or product flow rates. This includes values that were originally specifications but were replaced, thereby becoming initial estimates. It is your responsibility to ensure that you have entered a reasonable set of operating conditions (initial estimates) and specifications (Basic Column Parameters) that permit solution of the column. There are obviously many combinations of column configurations and specifications that makes convergence difficult or impossible. Although all these different conditions could not possibly be covered here, some of the more frequent problems are discussed in the following sections.

2.9.1 Heat and Spec Errors Fail to Converge

This is by far the most frequent situation encountered when a column is unable to satisfy the allowable tolerance. The following section gives the most common ailments and remedies.

Poor Initial Estimates

Initial estimates are important only to the extent that they provide the initial starting point for the tower algorithm. Generally, poor guesses simply cause your tower to converge more slowly. However, occasionally the effect is more serious. Consider the following:

- Check product estimates using approximate splits. A good estimate for the tower overhead flow rate is to add up all the components in your feed which are expected in the overheads, plus a small amount of your heavy key component. If the tower starts with extremely high errors, check to see that the overhead estimate is smaller than the combined feed rates.
- Poor reflux estimates usually do not cause a problem except in very narrow boiling point separations. Better estimates are required if you have high column liquid rates relative to vapour rates, or vice versa.
- Towers containing significant amounts of inert gases (for example H_2, N_2, and so forth), require better estimates of overhead rates to avoid initial bubble point problems. A nitrogen rejection column is a good example.

To see the initial estimates, click the View Initial Estimates button on the Monitor page of the column property view.

Input Errors

It is good practice to check all of your input just before running your column to ensure that all your entries, such as the stage temperatures and product flow rates, appear reasonable:

- Check to ensure that your input contains the correct values and units. Typical mistakes are entering a product flow rate in moles/hr when you really meant to enter it in barrels/day, or a heat duty in BTU/hr instead of E+06 BTU/hr.
- When specifying a distillate liquid rate, make sure you have specified the Distillate rate for the condenser, not the Reflux rate.
- If you change the number of trays in the column, make sure you have updated the feed tray locations, pressure specifications, and locations of other units such as side exchangers on the column.
- If the tower fails immediately, check to see if all of your feeds are known, if a feed was entered on a non-existent tray, or if a composition specification was mistakenly entered for a zero component.

Clicking the Input Summary button on the Monitor page of the column property view displays the column input in the Trace Window.

Incorrect Configuration

For more complex tower configurations, such as crude columns, it is more important that you always review your input carefully before running the tower. It is easy to overlook a stripping feed stream, side water draw, pump around or side exchanger. Any one of these omissions can have a drastic effect on the column performance. As a result, the problem is not immediately obvious until you have reviewed your input carefully or tried to change some of the specifications.

- Check for trays which have no counter-current vapour-liquid traffic. Examples of this are having a feed stream on a tray that is either below the top tray of an un-refluxed tower or a tower without a top lean oil feed, or placing a feed stream above the bottom stage of a tower that does not have a bottom reboiler or a stripping feed stream below it. In both cases the trays above or below the feed tray become single phase. Since they do not represent any equilibrium mass transfer, they should be removed or the feed should be moved. The tower cannot converge with this configuration.

- The tower fails immediately if any of the sidestrippers do not have a stripping feed stream or a reboiler. If this should occur, a message is generated stating that a reboiler or feed stream is missing in one of the sidestrippers.
- Make sure you have installed a side water draw if you have a steam-stripped hydrocarbon column with free water expected on the top stage.
- Regardless of how you have approached solving crude columns in the past, try to set up the entire crude column with your first run, including all the side strippers, side exchangers, product side draws, and pump arounds attached. Difficulties arise when you try to set up a more simplified tower that does not have all the auxiliary units attached to the main column, then assign product specs expected from the final configuration.

Impossible Specifications

Impossible specifications are normally indicated by an unchanging heat and spec error during the column iterations even though the equilibrium error is approaching zero. To get around this problem you have to either alter the column configuration or operating pressure or relax/change one of the product specifications.

- You cannot specify a temperature for the condenser if you are also using subcooling.
- If you have zero liquid flows in the top of the tower, either your top stage temperature spec is too high, your condenser duty is too low, or your reflux estimate is too low.
- If your tower shows excessively large liquid flows, either your purity specs are too tight for the given number of trays or your Cooler duties are too high.
- Dry trays almost always indicate a heat balance problem. Check your temperature and duty specifications. There are a number of possible solutions: fix tray traffic and let duty vary; increase steam rates; decrease product makes; check feed temperature and quality; check feed location.
- A zero product rate could be the result of an incorrect product spec, too much heat in the column which eliminates internal reflux, or the absence of a heat source under a total draw tray to produce needed vapour.

Conflicting Specifications

This problem is typically the most difficult to detect and correct. Since it is relatively common, it deserves considerable attention.

- You cannot fix all the product flow rates on a tower.
- Avoid fixing the overhead temperature, liquid and vapour flow rates because this combination offers only a very narrow convergence envelope.
- You cannot have subcooling with a partial condenser.
- A cut point specification is similar to a flow rate spec; you cannot specify all flows and leave one unspecified and then specify the cut point on that missing flow.
- Only two of the three optional specifications on a pump around can be fixed. For example, duty and return temperature, duty and pump around rate, and so forth.
- Fixing column internal liquid and vapour flows, as well as duties can present conflicts since they directly affect each other.
- The bottom temperature spec for a non-reboiled tower must be less than that of the bottom stage feed.
- The top temperature for a reboiled absorber must be greater than that of the top stage feed unless the feed goes through a valve.
- The overhead vapour rate for a reboiled absorber must be greater than the vapour portion of the top feed.

Heat and Spec Error Oscillates

While less common, this situation can also occur. It is often caused by poor initial estimates. Check for:

- Water condensation or a situation where water alternately condenses and vapourizes.
- A combination of specifications that do not allow for a given component to exit the column, causing the component to cycle in the column.
- Extremely narrow boiling point separations can be difficult since a small step change can result in total vapourization. First, change the specifications so that the products are not pure components. After convergence, reset the specifications and restart.

2.9.2 Equilibrium Error Fails to Converge

This is almost always a material balance problem. Check the overall balance.

- Check the tower profile. If the overhead condenser is very cold for a hydrocarbon-steam column, you need a water draw.

  Normally, a side water draw should be added for any stage below 200^{o}F.
- If the column almost converges, you may have too many water draws.

2.9.3 Equilibrium Error Oscillates

Refer to **Section 2.4.2 - Parameters Tab** for more information

This generally occurs with non-ideal towers, such as those with azeotropes. Decreasing the damping factor or using adaptive damping should correct this problem.

2.10 References

[1] Sneesby, Martin G., Simulation and Control of Reactive Distillation, Curtin University of Technology, School of Engineering, March 1998.

[2] Henry, Kister., Distillation Design, (1992), pp 497-499.

3 Electrolyte Operations

3.1 Introduction

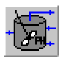
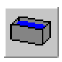
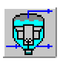
Most HYSYS unit operations can be used when working with the OLI Electrolyte property package.

Press F4 to open the Object Palette. The Object Palette shows the unit operations available in OLI Electrolyte property package by active icons.

The following HYSYS unit operations are not available in the OLI Electrolyte property package:

- Pipe Segment
- Reactors
- Short Cut Column
- Three Phase Distillation
- Compressible Gas Pipe

In addition to the typical HYSYS unit operations, three new electrolyte simulations, specific to OLI Electrolyte property package have been added. The table below describes the three new electrolyte simulations.:

Operation	Icon	Description
Neutralizer		Neutralizer operation is used to control PH value for a process material stream.
Precipitator		Precipitator operation is used to achieve a specified aqueous ionic species concentration in its product stream.
Crystalizer		Crystallizer operation is used to estimate and control solid concentration in a product stream.

The electrolyte operations are only available if your case is an electrolyte system (the selected fluid package must support electrolyte).

3.1.1 Adding Electrolyte Operations

There are two ways you can add an electrolyte operation to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

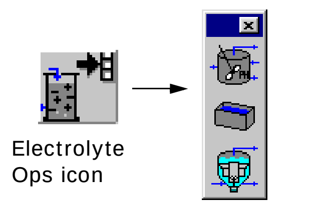
   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Electrolyte Equipment** radio button.
3. From the list of available unit operations, select the electrolyte operation you want.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.
2. Click the **Electrolyte Ops** icon. The electrolyte object palette appears.

Figure 3.1



3. Double-click the electrolyte operation you want.

The property view for the selected electrolyte operation appears.

The following sections describe the function of each electrolyte unit operation.

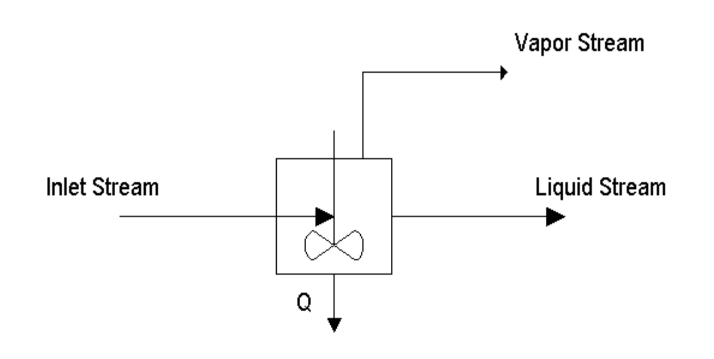
3.2 Crystalizer Operation

The Crystallizer operation models the crystallization of a fully defined inlet stream to attain a specified amount of selected solids concentration that is present in the effluent. The Crystalizer operation contains four tabs: Design, Rating, Worksheet, and Dynamics.

Theory

The figure below represents the crystallizer model. A Crystallizer has a product stream that contains liquid and solid. By adjusting the operation condition like Crystallizer temperature and pressure or heat duty, the amount of solid or solid component product in the liquid stream can be controlled or estimated.

Figure 3.2



The Crystallizer vessel is modeled as a perfect mixing in HYSYS. Heat can be added or removed from the Crystallizer, and a simple constant duty model is assumed.

Boundary Condition

Since the electrolyte flow sheet implements a forward calculation only, the Crystallizer does not solve until the Inlet Stream is defined. If the energy stream is not specified, the crystallizer is treated as an adiabatic one. You must specify two of the following to define the boundary condition for crystallizer solver to proceed:

- T. Crystallizer temperature
- P or DeltP. Crystallizer's pressure or pressure drop
- E. Heat Duty
- F_{cry}. Crystal product flow rate (total or a specific component)
- F_{vap}. Vapor flow

Equations

The crystallizer solves under the constraint of mass and energy balance equations:

$$E_{\text{product stream}} + E_{\text{vapour stream}} = E_{\text{inlet stream}} + E_{\text{duty}} \tag{3.1}$$

$$M_{\text{product stream}} + M_{\text{vapour stream}} = M_{\text{inlet stream}} \tag{3.2}$$

with the target solid equation:

$$F_{solid}(\text{product stream}) - F_{solid}(\text{specified}) = 0 \tag{3.3}$$

where:

F_{solid}(product stream) = *solid flow rate in the outlet liquid stream*

F_{solid}(specified) = *desired solid flow rate in the outlet liquid stream*

E = energy/heat transfer rate

M = mass flow rate

3.2.1 Design Tab

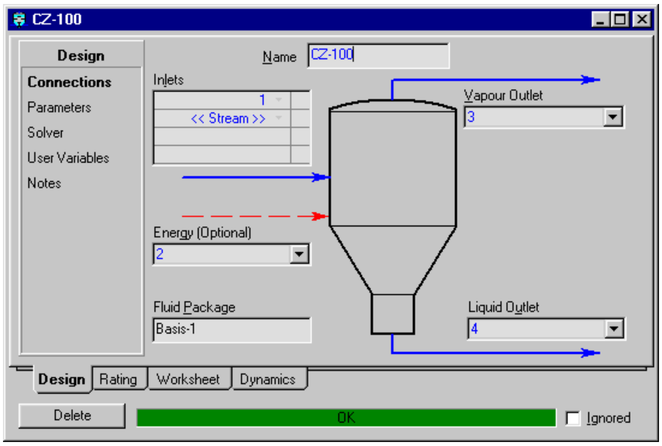
The Design tab contains the following pages:

- Connections
- Parameters
- Solver
- User Variables
- Notes

Connections Page

You can specify the inlet stream, outlet stream, and energy stream on the Connections page.

Figure 3.3



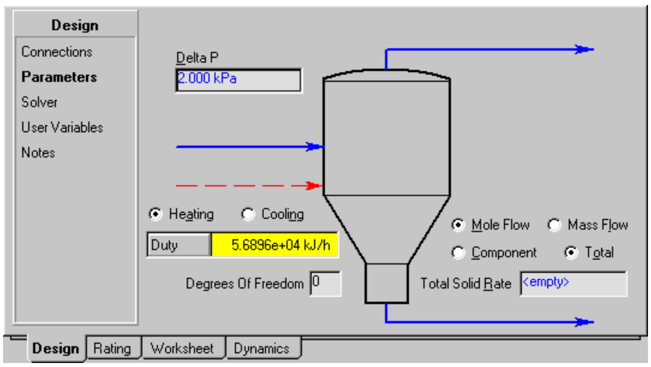
Object	Description
Name	You can change the name of the operation by typing a new name in the field.
Inlet	You can enter one or more inlet streams in this table, or use the drop-down list to select the streams you want.
Vapour Outlet	You can enter the name of the vapour product stream or use the drop-down list to select a pre-defined stream.
Liquid Outlet	You can enter the name of the product stream in this field or use the drop-down list to select a pre-defined stream.

Object	Description
Energy (Optional)	You can add an energy stream to the operation by selecting an energy stream from the drop-down list or typing the name for a new energy stream.
Fluid Package	Displays the fluid package currently being used by the operation. You can select a different fluid package from the drop-down list.

Parameters Page

On the Parameters page, you can specify the pressure drop and solid output flow rate.

Figure 3.4



The flow rate of crystal product depends on the solubility of the product at the crystallizer's operation condition.

The four radio buttons allow you to control the specified solid output in the liquid stream by crystallization operation:

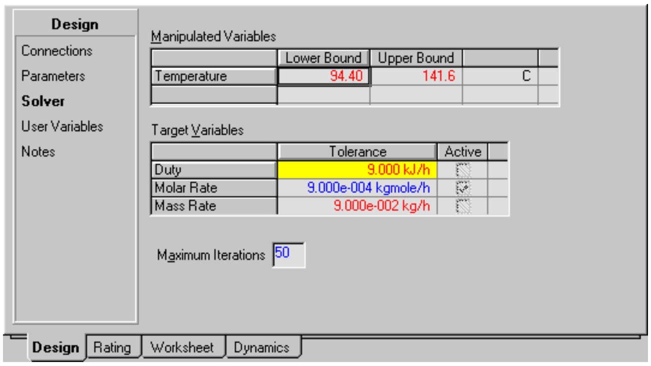
- **Mole Flow**. Select this radio button to specify the flow rate value in mole basis.
- **Mass Flow**. Select this radio button to specify the flow rate value in mass basis.
- **Component**. Select this radio button to control a specified solid component in the operation.
- **Total**. Select this radio button to control the total solid flow rate in the liquid stream.

This page also displays the degrees of freedom for the operation at the current setting.

Solver Page

On the Solver page, you can specify the upper and lower bounds of the manipulated variable, the tolerance of specified variable, and the maximum iterations/steps of calculations the solver performs before stopping.

Figure 3.5



Crystallizer operates on various boundary conditions. The following table lists all the possible options. As soon as the operation condition (as listed in the Specified Variables column) is known, the crystallizer will start to solve. The Crystallizer Calculates column lists some of the calculation variables for the operation.

Specified Variables	Crystallizer Calculates
Temperature & Pressure	Heat Duty, Crystal product flow rate, Vapour flow rate
Temperature & Heat Duty	Pressure, Crystal product flow rate, Vapor flow rate
Temperature & Crystal product flow rate	Pressure, Heat Duty, Vapor flow rate
Temperature & Vapour flow rate	Pressure, Crystal product flow rate, Heat Duty
Pressure & Heat Duty	Temperature, Crystal product flow rate, Vapor flow rate
Pressure & Crystal product flow rate	Temperature, Heat Duty, Vapor flow rate
Pressure & Vapour flow rate	Temperature, Crystal product flow rate, Heat Duty

The bounds for the Manipulated Variables and tolerances for the Target Variables are shown on the Solver tab and are user-modifiable. As well, the Active status for the Manipulated Variable used by the solver is shown. However, this flag is meant for displaying information only thus cannot be changed.

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

3.2.2 Rating Tab

Crystalizer operation currently does not support any rating calculations.

3.2.3 Worksheet Tab

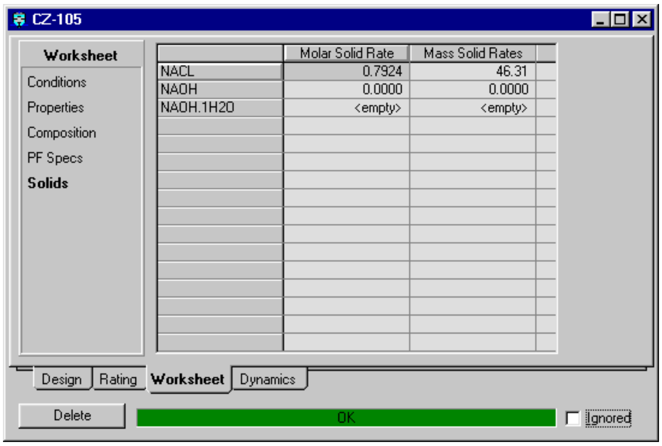
Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the Crystallizer.

The PF Specs page is relevant to dynamics cases only.

The Crystallizer Worksheet tab also has one extra page called the Solids page. On the Solids page, you can view the precipitate molar and mass flow rates.

Figure 3.6



3.2.4 Dynamic Tab

Crystalizer operation currently does not support dynamic mode.

3.3 Neutralizer Operation

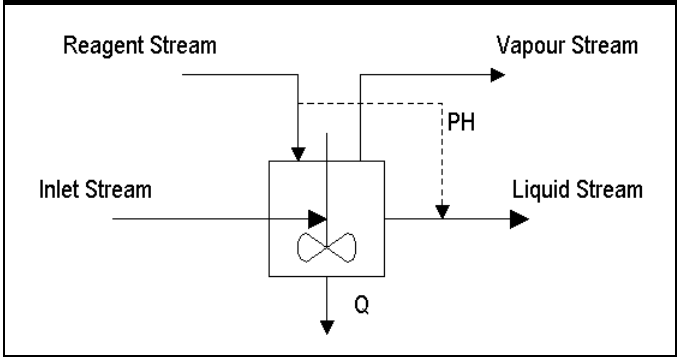
The Neutralizer operation models the neutralization of a fully defined inlet stream, and allows you to adjust the pH value in the effluent stream. The Neutralizer property view contains four tabs:

- Design
- Rating
- Worksheet
- Dynamics

Theory

The figure below represents the neutralizer model. Through adjusting the Reagent Stream variables (flow rate), the PH value for the targeting stream (Liquid Stream) could be controlled at the level as required.

Figure 3.7



- **Inlet Stream**. At least one inlet stream.
- **Reagent Stream**. Reagent stream must be a free stream, that is, not attached to any other unit operations.
- **Product Stream**. A Neutralizer has two product streams, a vapour stream and a liquid stream. The liquid stream controls the pH value.
- **pH**. The liquid stream’s pH value that is to be controlled must fall in the range between the pH values of the Reagent and inlet streams to guarantee the solution.
- **Q**. The energy stream is optional. When no energy stream is attached, an adiabatic operation is assumed.

The Neutralizer vessel is modeled as perfect mixing. Heat can be added or removed from the Neutralizer, and a simple constant duty model is assumed.

Boundary Condition

Since the electrolyte flow sheet implements a forward calculation only, the Neutralizer does not solve until both inlet and Reagent streams are defined.

If the energy stream is not specified, the neutralizer is treated as an adiabatic one. If the energy stream is specified, you must specify either the Neutralizer temperature or the duty of the energy stream.

Pressure drop of the neutralizer must be specified or can be calculated out from the inlet and product streams.

Solving Options

The Neutralizer has two different solving options, depending on what you specify.

Option 1 (Targeting pH Value is Not Specified)

If the targeting pH value is not specified, the Neutralizer operates as a mixer for the inlet and Reagent streams. The product stream accepts the mixed result as is.

Option 2 (Targeting pH Value is Specified)

If the targeting pH value is specified, the flow rate of the Reagent stream must be left unspecified. The Reagent stream is used as an adjusting variable for neutralizer solver to search for a solution to meet the targeting pH value at the outlet stream.

Equations

The Neutralizer solves under the constraint of the following equations.

$$\mathrm{pH}_{\text{product stream}} - \mathrm{pH}_{\text{specified}} = 0 \tag{3.4}$$

$$\mathrm{pH}_{\text{specified}} \subset \{\mathrm{pH}_{\text{inlet stream}},\ \mathrm{pH}_{\text{Reagent stream}}\} \tag{3.5}$$

$$E_{\text{product stream}} = E_{\text{inlet stream}} + E_{\text{Reagent stream}} + E_{\text{duty}} \tag{3.6}$$

$$M_{\text{product stream}} = M_{\text{inlet stream}} + M_{\text{Reagent stream}} \tag{3.7}$$

where:

E = energy/heat transfer rate

M = mass flow rate

3.3.1 Design Tab

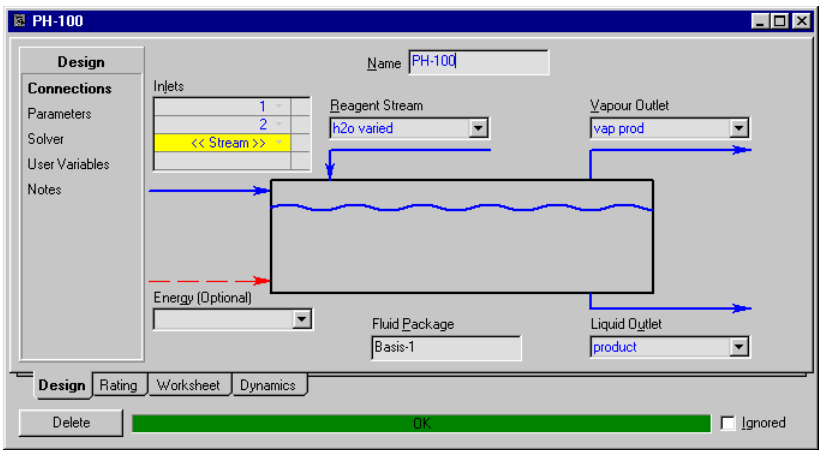
The Design tab contains the following pages:

- Connections
- Parameters
- Solver
- User Variables
- Notes

Connections Page

You can specify the inlet stream, outlet stream, and energy stream on the Connections page.

Figure 3.8

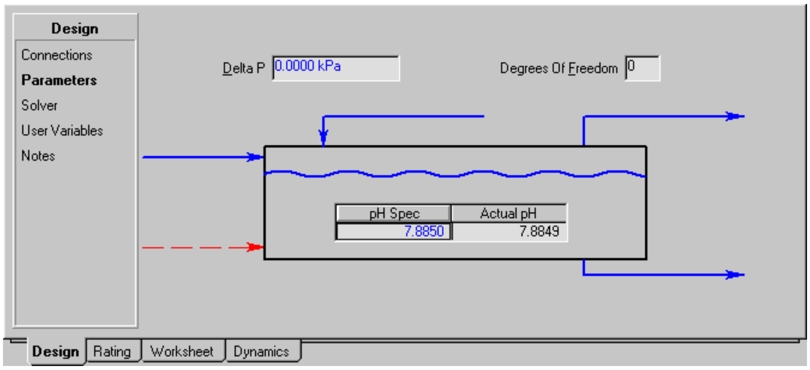


Object	Description
Name	You can change the name of the operation by typing a new name in the field.
Inlet	You can enter one or more inlet streams in this table, or use the drop-down list to select the streams you want.
Reagent Stream	You can enter a name for the reagent stream or use the drop-down list. Reagent stream must be a free stream, that is, not attached to any other unit operations.
Vapour Outlet	You can type the name of the vapour product stream or use the drop-down list to select a pre-defined stream.
Liquid Outlet	You can type the name of the product stream in this field or use the drop-down list to select a pre-defined stream.
Energy (Optional)	You can add an energy stream to the operation by selecting an energy stream from the drop-down list or typing the name for a new energy stream.
Fluid Package	Displays the fluid package currently being used by the operation. You can select a different fluid package from the drop-down list.

Parameters Page

On the Parameters page, you can specify the pressure drop and an initial pH value. This page also displays the degrees of freedom for the operation at the current setting, and the actual pH balance in the operation when the operation reaches a solution.

Figure 3.9



Object	Description
Delta P	You must specify the pressure drop for the Neutralizer or specify inlet and product streams with known pressure.
pH Spec	You can specify the product stream's pH value in this field. The pH value that is to be controlled must fall in the range between the pH values of the Reagent and Inlet Streams for calculations to converge.

The pH value in a solution is defined in a mathematical format:

$$pH = -\log_{10}[H^+] \tag{3.8}$$

where:

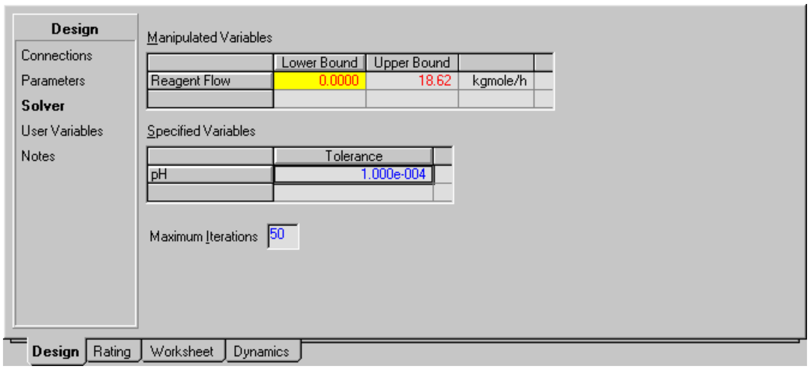
$[H^+]$ = *concentration of H^+ in a solution, $^{mol}/l$*

According to **Equation (3.5)**, the pH (specified) value must be specified between the pH values of the inlet and the Reagent streams. An adjustment of Reagent Stream's variables, for example, temperature, pressure, and compositions, can bracket the pH (specified) value to meet the constraint **Equation (3.5)**. As soon as the specified pH value is bracketed according to **Equation (3.5)**, the pH value of the product stream in **Equation (3.4)** can be obtained by adjusting the flow rate of the Reagent stream.

Solver Page

On the Solver page, you can specify the upper and lower bounds of the manipulated variable, the tolerance of specified variable, and the maximum iterations/steps of calculations the solver performs before stopping.

Figure 3.10



Currently only the flow rate of a defined Reagent stream is used as an adjustable variable to the solver. Here a defined Reagent stream means that the stream can be flashed to get a solution with the specified variables meeting the degree of freedom. According to HYSYS, a defined stream can have the following variables:

- T. Stream Temperature
- P. Stream Pressure
- F. Stream Flow Rate
- x. Stream Component Compositions
- H. Stream Enthalpy
- V. Stream Vapor Fraction

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

3.3.2 Rating Tab

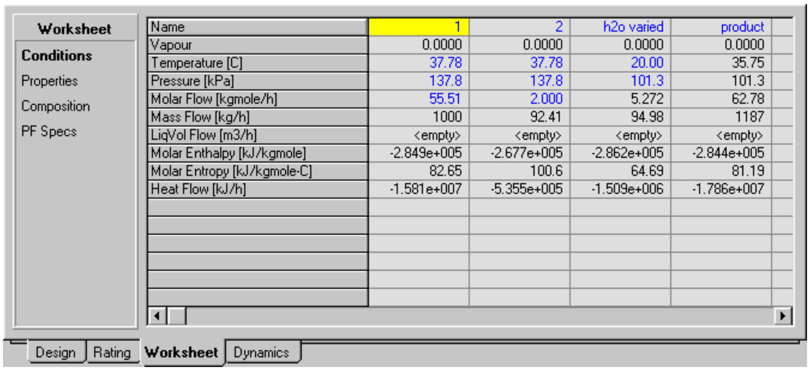
Neutralizer operation currently does not support any rating calculations.

3.3.3 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the Neutralizer.

Figure 3.11



Name	1	2	h2o varied	product
Vapour	0.0000	0.0000	0.0000	0.0000
Temperature [C]	37.78	37.78	20.00	35.75
Pressure [kPa]	137.8	137.8	101.3	101.3
Molar Flow [kgmole/h]	55.51	2.000	5.272	62.78
Mass Flow [kg/h]	1000	92.41	94.98	1187
LiqVol Flow [m3/h]	<empty>	<empty>	<empty>	<empty>
Molar Enthalpy [kJ/kgmole]	-2.849e+005	-2.677e+005	-2.862e+005	-2.844e+005
Molar Entropy [kJ/kgmole-C]	82.65	100.6	64.69	81.19
Heat Flow [kJ/h]	-1.581e+007	-5.355e+005	-1.509e+006	-1.786e+007

The PF Specs page is relevant to dynamics cases only.

3.3.4 Dynamic Tab

Neutralizer operation currently does not support dynamic mode.

3.4 Precipitator Operation

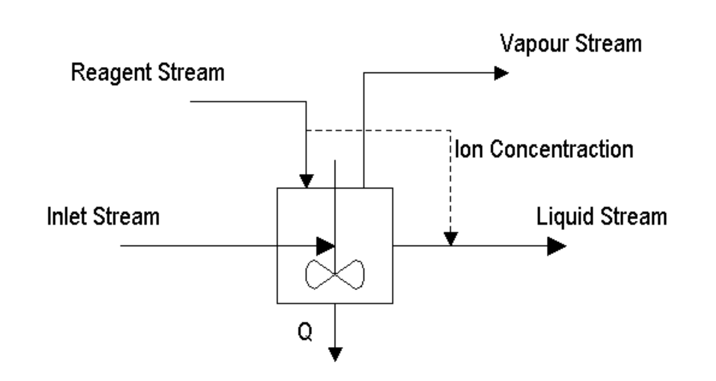
The Precipitator models the precipitation of a selected ion in a stream entering the operation to achieve a specified target concentration in the effluent stream. The Precipitator operation contains four tabs:

- Design
- Rating
- Worksheet
- Dynamics

Theory

The figure below represents the precipitator model.

Figure 3.12



Through adjusting the flow rate of the Reagent stream, the concentration of the targeting ion could be controlled at the desired level as you require in the outlet stream. To ensure that the Precipitator functions properly, the ions in the Reagent stream must be capable of reacting with the target ion under the specified operation condition. The formation of a precipitate in the outlet stream reduces the target ion concentration that entered the operation in the inlet stream.

- **Inlet Stream**. At least one inlet stream.

- **Reagent Stream**. Reagent stream must be a free stream, that is, not attached to any other unit operations.
- **Liquid Stream**. A Precipitator must have one liquid stream (contains liquid and solid) that is a targeting stream for the control of ion concentration through precipitation.
- **Ion Concentration**. The product stream's ion concentration value can be controlled by dilution or precipitation.
- **Q**. The energy stream is optional.

The Precipitator is modeled as a perfect mixing in HYSYS. Heat can be added or removed from the precipitator through a duty stream, and a simple constant duty model is assumed.

Boundary Condition

Since the electrolyte flow sheet implements a forward calculation only, the Precipitator does not solve until both inlet and Reagent streams are defined. If the energy stream is not specified, the precipitator is treated as an adiabatic one. If the energy stream is specified, you must specify either the Precipitator temperature or the duty of the energy stream. Pressure drop of the Precipitator must be either specified or can be calculated from the inlet and product streams.

Solving Options

The Precipitator has two different solving options, depending on what you specify.

- Option 1 (Targeting Ionic Species Not Specified)
  If the targeting ionic species is not specified, the Precipitator simply mixes the inlet stream with the Reagent stream. The product stream accepts the mixed result as is.
- Option 2 (Targeting Ionic Species is Specified)
  If the targeting ionic species is specified for the control of its concentration, the flow rate of the Reagent stream is used as iterative variables for the precipitator solver to search for a solution.

Equations

The precipitator solves under the constraint of the following equations:

$$C_{\text{ion}}(\text{product stream}) < C_{\text{ion}}(\text{specified}) \tag{3.9}$$

$$E_{\text{product stream}} + E_{\text{vapour stream}} + E_{\text{duty}} = E_{\text{inlet stream}} + E_{\text{Reagent stream}} \tag{3.10}$$

$$M_{\text{product stream}} + M_{\text{vapour stream}} = M_{\text{inlet stream}} + M_{\text{Reagent stream}} \tag{3.11}$$

where:

C_{ion} = *concentration of the targeting ion species*

E = energy/heat transfer rate

M = mass flow rate

3.4.1 Design Tab

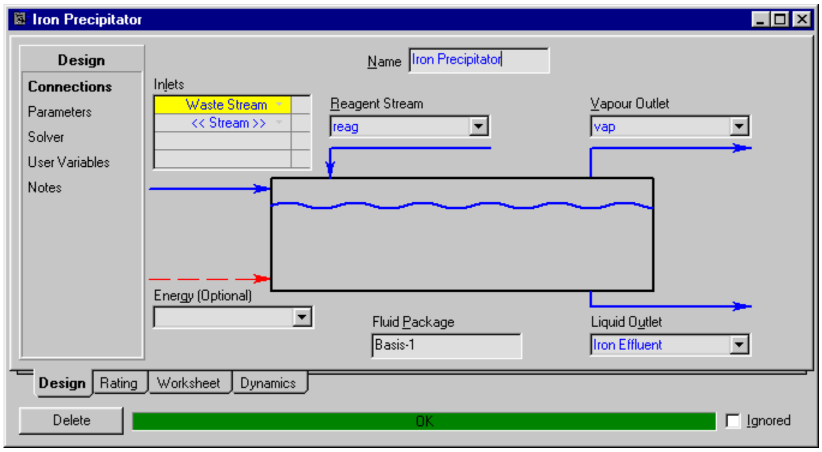
The Design tab contains the following pages:

- Connections
- Parameters
- Solver
- User Variables
- Notes

Connections Page

You can specify the inlet stream, outlet stream, and energy stream on the Connections page.

Figure 3.13

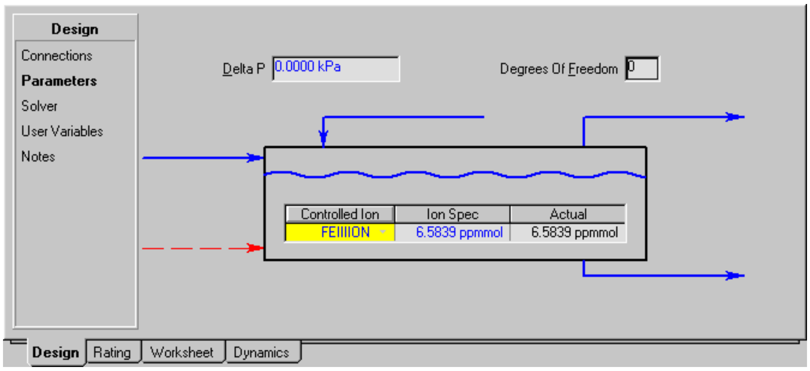


Object	Description
Name	You can change the name of the operation by typing a new name in the field.
Inlet	You can enter one or more inlet streams in this table, or use the drop-down list to select the streams you want.
Reagent Stream	You can enter a name for the reagent stream or use the drop-down list. Reagent stream must be a free stream, that is, not attached to any other unit operations.
Vapour Outlet	You can enter the name of the vapour product stream or use the drop-down list to select a pre-defined stream.
Liquid Outlet	You can enter the name of the product stream in this field or use the drop-down list to select a pre-defined stream.
Energy (Optional)	You can add an energy stream to the operation by selecting an energy stream from the drop-down list or typing the name for a new energy stream.
Fluid Package	Displays the fluid package currently being used by the operation. You can select a different fluid package from the drop-down list.

Parameters Page

On the Parameters page, you can specify the pressure drop, select the ion to be controlled, and specify the ion concentration in the liquid stream. This page also displays the degrees of freedom for the operation at the current setting, and the actual ion concentration value in the operation when the operation has reached a solution.

Figure 3.14

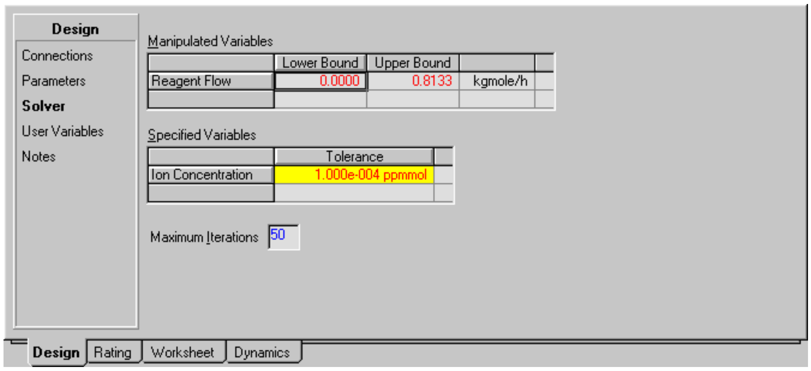


Object	Description
Delta P	You must specify the pressure drop for the Precipitator or specify inlet and product streams with known pressure.
Controlled Ion	Select the ion component you want to control from the drop-down list, or type the name of the ion component in the field.
Ion Spec	The concentration of ion from the inlet stream can be controlled via the following exercises: • **Dilution**. If the mixing of reagent and inlet streams does not produce the ion to be controlled and the ion concentration in the Reagent stream is less than that in the inlet stream, an increase of flow rate of the Reagent stream can achieve the target. In this case, the Chemistry Model does not have to include Solid. • **Precipitation**. Form precipitator by mixing inlet and Reagent streams. The change of Regent stream variables: temperature, pressure, flow rate or composition may achieve the target. To form precipitator, OLI chemistry model must include Solid.

Solver Page

On the Solver page, you can specify the upper and lower bounds of the manipulated variable, the tolerance of specified variable, and the maximum iterations/steps of calculations the solver performs before stopping.

Figure 3.15



Currently, the flow rate of the Reagent stream is the manipulated variable used by the precipitator solver to search for a solution.

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

3.4.2 Rating Tab

Precipitator operation currently does not support any rating calculations.

3.4.3 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the Precipitator.

The PF Specs page is relevant to dynamics cases only.

3.4.4 Dynamic Tab

Precipitator operation currently does not support dynamic mode.

4 Heat Transfer Operations

4.1 Air Cooler

The Air Cooler unit operation uses an ideal air mixture as a heat transfer medium to cool (or heat) an inlet process stream to a required exit stream condition. One or more fans circulate the air through bundles of tubes to cool process fluids. The air flow can be specified or calculated from the fan rating information. The Air Cooler can solve for many different sets of specifications including the:

- Overall heat transfer coefficient, UA
- Total air flow
- Exit stream temperature

4.1.1 Theory

Steady State

The Air Cooler uses the same basic equation as the Heat Exchanger unit operation, however, the Air Cooler operation can calculate the flow of air based on the fan rating information.

The Air Cooler calculations are based on an energy balance between the air and process streams. For a cross-current Air Cooler, the energy balance is calculated as follows:

$$M_{air}(H_{out} - H_{in})_{air} = M_{process}(H_{in} - H_{out})_{process} \qquad (4.1)$$

where:

M_{air} = *air stream mass flow rate*

$M_{process}$ = *process stream mass flow rate*

H = *enthalpy*

The Air Cooler duty, Q, is defined in terms of the overall heat transfer coefficient, the area available for heat exchange, and the log mean temperature difference:

$$Q = -UA\Delta T_{LM}F_t \tag{4.2}$$

where:

U = overall heat transfer coefficient

A = surface area available for heat transfer

ΔT_{LM} *= log mean temperature difference (LMTD)*

F_t *= correction factor*

The LMTD correction factor, F_t, is calculated from the geometry and configuration of the Air Cooler.

ACOL Functionality

In Steady State mode, you can also access certain ACOL functions on the HTFS-ACOL tab.

You must install and license ACOL 6.4 before you can access the ACOL functions.

Dynamic

In dynamics, the Air Cooler tube is capable of storing inventory like other dynamic unit operations. The direction of the material flowing through the Air Cooler operation is governed by the pressures of the surrounding unit operations.

Heat Transfer

The Air Cooler uses the same basic energy balance equations as the Heat Exchanger unit operation. The Air Cooler calculations are based on an energy balance between the air and process streams.

For a cross-current Air Cooler, the energy balance is shown as follows:

$$M_{process}(H_{in}-H_{out})_{process}-M_{air}(H_{in}-H_{out})_{air} = \rho\frac{d(VH_{out})_{process}}{dt} \qquad (4.3)$$

where:

M_{air} = *air stream mass flow rate*

$M_{process}$ = *process stream mass flow rate*

ρ = *density*

H = *enthalpy*

V = *volume of Air Cooler tube*

Pressure Drop

The pressure drop of the Air Cooler can be determined in one of two ways:

- Specify the pressure drop.
- Define a pressure flow relation in the Air Cooler by specifying a k-value.

If the pressure flow option is chosen for pressure drop determination in the Air Cooler, a *k value* is used to relate the frictional pressure loss and flow through the exchanger. This relation is similar to the general valve equation:

$$flow = \sqrt{density}\times k\sqrt{P_1-P_2} \qquad (4.4)$$

The general flow equation uses the pressure drop across the Heat Exchanger without any static head contributions. The quantity, P_1 - P_2, is defined as the frictional pressure loss which is used to “size” the Air Cooler with a *k value*.

Dynamic Specifications

In general, three specifications are required by HYSYS in order for the Air Cooler unit operation to fully solve:

Dynamic Specifications	Description
Overall UA	The product of the Overall Heat Transfer Coefficient, and the total area available for heat transfer. The Overall UA must be specified in Dynamic mode. You can specify the value of UA on the Parameters page of the Design tab.
Fan Rating Information	You must specify the following information on the Sizing page of the Rating tab: • Demanded Speed • Design Speed • Design Flow • Max Acceleration (optional) or • Current Air Flow
Pressure Drop	Either specify an Overall Delta P or an Overall K-value for the Air Cooler. These pressure drop specifications can be made on the Specs page of the Dynamics tab.

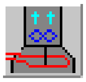
4.1.2 Air Cooler Property View

Add an Air Cooler to your simulation by doing the following:

1. Select **Flowsheet | Add Operation** command from the menu bar (or press **F12**). The UnitOps property view appears.
2. Click the **Heat Transfer Equipment** radio button.
3. From the list of available unit operations, select Air Cooler.
4. Click the **Add** button.

OR

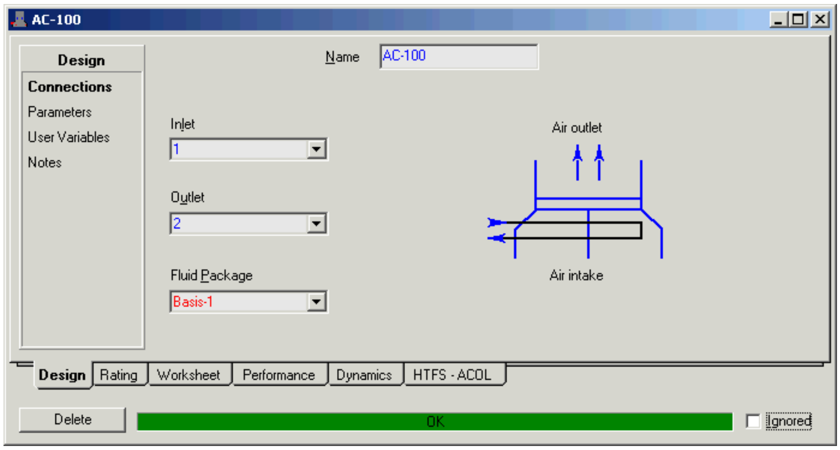
1. Select **Flowsheet | Palette** command from the menu bar (or press **F4**). The Object Palette appears.
2. Double-click the **Air Cooler** icon.



Air Cooler icon

The Air Cooler property view appears.

Figure 4.1



4.1.3 Design Tab

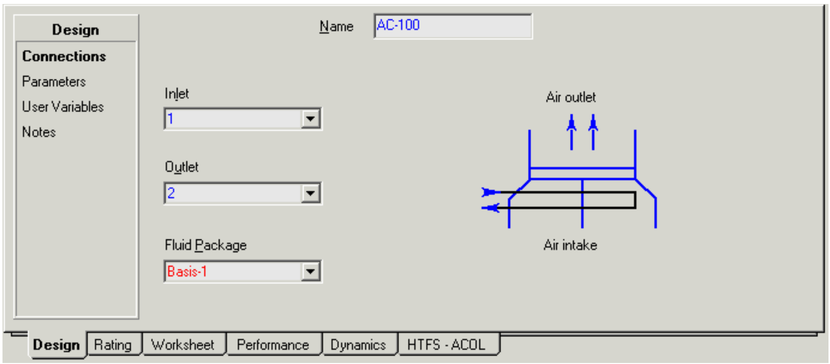
The Design tab contains the following pages:

- Connections
- Parameters
- User Variables
- Notes

Connections Page

On the Connections page, you can specify the feed and product streams attached to the Air Cooler. You can change the name of the operation in the Name field.

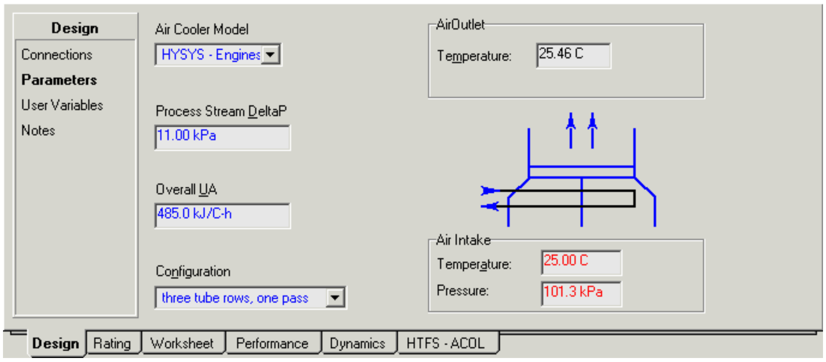
Figure 4.2



Parameters Page

On the Parameters page, the following information appears:

Figure 4.3



Parameters	Description
Air Cooler Model	Allows you to select HYSYS-Engines or HTFS-Engines. The HTFS-Engines options appears only if you have ACOL6.4 installed and licensed. The HTFS-Engines option allows you to access ACOL functions on the HTFS-ACOL tab.
Process Stream Delta P	Allows you to specify the pressure drops (DP) for the process stream side of the Air Cooler. The pressure drop can be calculated if both the inlet and exit pressures of the process stream are specified. There is no pressure drop associated with the air stream. The air pressure through the Cooler is assumed to be atmospheric.
Overall UA	Contains the value of the Overall Heat Transfer Coefficient multiplied with the Total Area available for heat transfer. The Air Cooler duty is proportional to the log mean temperature difference, where UA is the proportionality factor. The UA can either be specified or calculated by HYSYS.
Configuration	Displays the possible tube pass arrangements in the Air Cooler. There are seven different Air Cooler configurations to choose from. HYSYS determines the correction factor, Ft, based on the selected Air Cooler configuration.
Air Intake / Outlet Temperatures	The inlet and exit air stream temperatures can be specified or calculated by HYSYS.
Air Intake Pressure	The inlet air stream pressure has a default value of 1 atm.

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation or the simulation case in general.

4.1.4 Rating Tab

The Rating tab allows you to specify the fan rating information. The steady state and dynamic Air Cooler operations share the same fan rating information.

> **In dynamics, the air flow must be calculated using the fan rating information.**

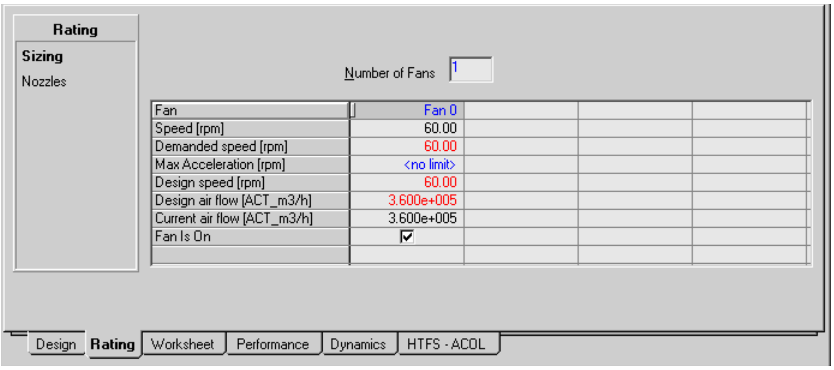
The Rating tab contains the following pages:

- Sizing page. The content of this page differs depending on which option you selected in the Air Cooler Model drop-down list on the Parameters page of the Design tab. If you selected HTFS-Engines, this page displays only one field: Air Mass Flow Rate.
- Nozzles page. This page appears only if the HYSYS Dynamics license is activated.

Sizing Page HYSYS-Engines

In the Sizing page, the following fan rating information appears for the Air Cooler operation when the HYSYS-Engines option is selected on the Parameters page of the Design tab.

Figure 4.4



Fan Data	Description
Number of Fans	Number of fans in the Air Cooler.
Speed	Actual speed of the fan in rpm (rotations per minute).
Demanded speed	Desired speed of the fan. • Steady State mode. The demanded speed is always equal the speed of the fan. The desired speed is either calculated from the fan rating information or user-specified. • Dynamic mode. The demanded speed should either be specified directly or from a Spreadsheet operation. If a control structure uses the fan speed as an output signal, it is the demanded speed which should be manipulated.
Max Acceleration	Applicable only in Dynamic mode. It is the rate at which the actual speed moves to the demanded speed.
Design speed	The reference Air Cooler fan speed. It is used in the calculation of the actual air flow through the Cooler.
Design air flow	The reference Air Cooler air flow. It is used in the calculation of the actual air flow through the Cooler.
Current air flow	This can be calculated or user-specified. If the air flow is specified no other fan rating information needs to be specified.
Fan Is On	By default, this checkbox is selected. You have the option to turn on or off the air cooler as desired. When you clear the checkbox, the temperature of the outlet stream of the air cooler will be identical to that of the inlet stream. The **Fan Is On** checkbox has the same function as setting the Speed to 0 rpm.

The air flow through the fan is calculated using a linear relation:

$$Fan\ Air\ Flow = \frac{Speed}{Design\ Speed} \times Design\ Flow \tag{4.5}$$

In dynamic mode only, the actual speed of the fan is not always equal to the demanded speed. The actual fan speed after each integration time step is calculated as follows:

$$\begin{gathered} Actual\ Speed = (Max\ Acceleration)\Delta t + Actual\ Speed_o \\ \text{until } Actual\ Speed = Demanded\ Speed \end{gathered} \tag{4.6}$$

Each fan in the Air Cooler contributes to the air flow through the Cooler. The total air flow is calculated as follows:

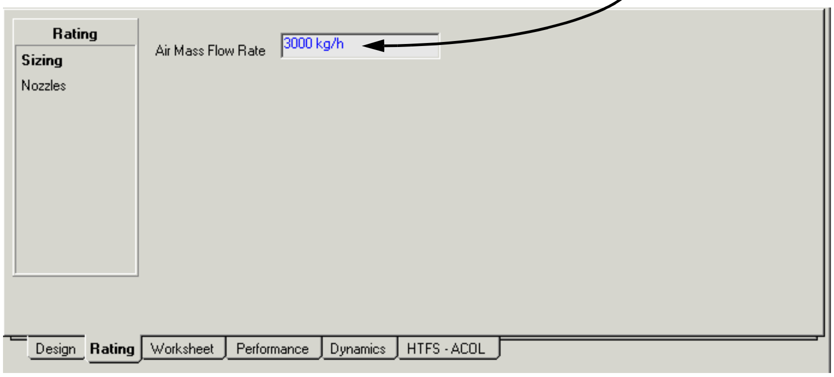
$$Total\ Air\ Flow = \sum Fan\ Air\ Flow \tag{4.7}$$

Sizing Page HTFS-Engines

The following page appears when the HTFS-Engines option is selected on the Parameters page of the Design tab.

Figure 4.5

Enter the total airflow for the air cooler in this field.



HYSYS air coolers can have multiple fans, and HYSYS calculates the airflow from the sum of the airflows of each fan.

When you select HYSYS-Engines on the Parameters page of the Design tab, HYSYS allows you to define the parameters for each fan on this page, however, you can only enter the total air mass flow rate for the air cooler.

Nozzles Page

Refer to **Section 1.3.6 - Nozzles Page** for more information.

The Nozzles page contains information regarding the elevation and diameter of the nozzles. The information provided in the Nozzles page is applicable only in Dynamic mode.

4.1.5 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the Air Cooler.

The PF Specs page is relevant to dynamics cases only.

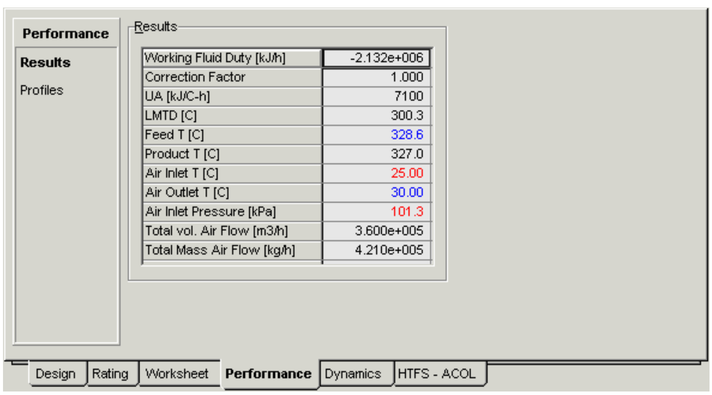
4.1.6 Performance Tab

The Performance tab contains pages that display the results of the Air Cooler calculations.

The Profiles page is relevant to dynamics cases only.

Results Page

Figure 4.6



The information from the Results page is shown as follows:

Results	Description
Working Fluid Duty	This is defined as the change in duty from the inlet to the exit process stream: $$H_{process,\ in} + Duty = H_{process,\ out}$$
LMTD Correction Factor, Ft	The correction factor is used to calculate the overall heat exchange in the Air Cooler. It accounts for different tube pass configurations.
UA	The product of the Overall Heat Transfer Coefficient, and the Total Area available for heat transfer. The UA can either be specified or calculated by HYSYS.
LMTD	The LMTD is calculated in terms of the temperature approaches (terminal temperature difference) in the exchanger, using the following uncorrected LMTD equation: $$\Delta T_{LM} = \frac{\Delta T_1 - \Delta T_2}{\ln(\Delta T_1 / (\Delta T_2))}$$ *where:* $$\Delta T_1 = T_{hot,\ out} - T_{cold,\ in}$$ $$\Delta T_2 = T_{hot,\ in} - T_{cold,\ out}$$
Inlet/Outlet Process Temperatures	The inlet and outlet process stream temperatures can be specified or calculated in HYSYS.
Inlet/Outlet Air Temperatures	The inlet and exit air stream temperatures can be specified or calculated in HYSYS.
Air Inlet Pressure	The inlet air stream pressure has a default value of 1 atm.
Total Air flow	The total air flowrate appears in volume and mass units.

4.1.7 Dynamics Tab

The Dynamics tab contains the following pages:

- Model
- Specs
- Holdup
- Stripchart

In dynamics, the air flow must be calculated using the fan rating information.

If you are working exclusively in Steady State mode, you are not required to change any of the values on the pages accessible through this tab.

Model Page

The Model page allows you to define how UA is defined in Dynamic mode. The value of *UA* is calculated as follows:

$$UA_{dynamic} = F \times UA_{steadystate} \tag{4.8}$$

where:

$UA_{steadystate}$ = UA value entered on the Parameters page of the Design tab.

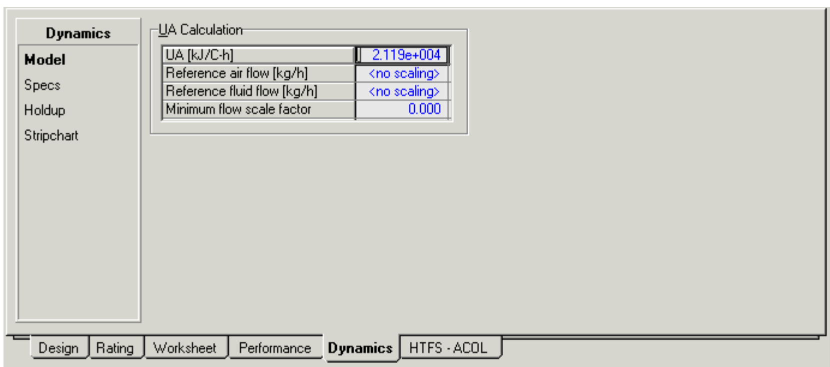
$$F = \frac{2 \times f1 \times f2}{(f1 + f2)} \quad \textit{the flow scale factor} \tag{4.9}$$

f1 = (mass flowrate / reference flowrate) ^0.8 *for air* (4.10)

f2 = (mass flowrate / reference flowrate) ^0.8 *for fluid* (4.11)

The Model page contains one group, the UA Calculation.

Figure 4.7



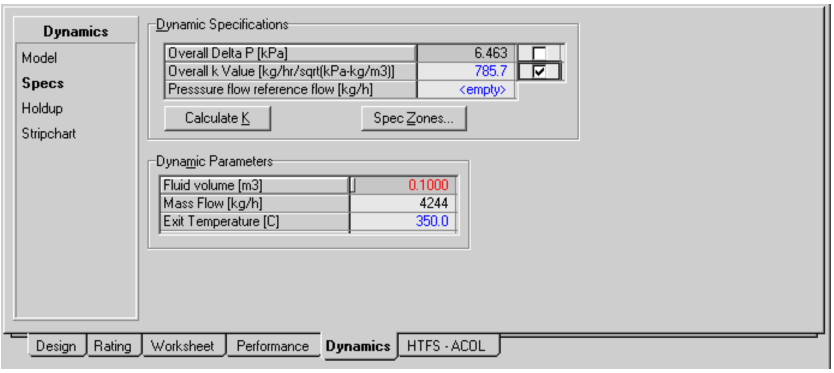
The group contains four fields, which are described in the table below.

Field	Description
UA	The steady state value of UA. This should be the same as the value entered on the Parameters tab.
Reference air flow	The reference flowrate for air. It is used to calculate the value of *f*1 as shown in **Equation (4.10)**.
Reference fluid flow	The reference flowrate for the fluid. It is used to calculate the value of *f*2 as shown in **Equation (4.11)**.
Minimum flow scale factor	The minimum scale factor used. If the value calculated by **Equation (4.9)** is smaller than this value, this value is used.

Specs Page

The Specs page contains information regarding the calculation of pressure drop across the Air Cooler:

Figure 4.8



You can specify how the pressure drop across the Air Cooler is calculated in the Dynamic Specifications group.

Dynamic Specifications	Description
Overall Delta P	A set pressure drop is assumed across the valve operation with this specification. The flow and the pressure of either the inlet or exit stream must be specified or calculated from other operations in the flowsheet. The flow through the valve is not dependent on the pressure drop across the Air Cooler. To use the overall delta P as a dynamic specification, select the corresponding checkbox. The Air Cooler operations, like other dynamic unit operations, should use the k-value specification option as much as possible to simulate actual pressure flow relations in the plant.
Overall k Value	The k-value defines the relationship between the flow through the Air Cooler and the pressure of the surrounding streams. You can either specify the k-value or have it calculated from the stream conditions surrounding the Air Cooler. You can “size” the Cooler with a k-value by clicking the Calculate K button. Ensure that there is a non zero pressure drop across the Air Cooler before the Calculate K button is clicked. To use the k-value as a dynamic specification, select the corresponding checkbox.
Pressure Flow Reference Flow	The reference flow value results in a more linear relationship between flow and pressure drop. This is used to increase model stability during startup and shutdown where the flows are low. If the pressure flow option is chosen the k value is calculated based on two criteria. If the flow of the system is larger than the k Reference Flow the k value remains unchanged. It is recommended that the k reference flow is taken as 40% of steady state design flow for better pressure flow stability at low flow range. If the flow of the system is smaller than the k Reference Flow the k value is given by: $k_{used} = k_{specified} \times Factor$ where Factor is determined by HYSYS internally to take into consideration the flow and pressure drop relationship at low flow regions.

The Dynamic Parameters group contains information about the holdup of the Air Cooler, which is described in the table below.

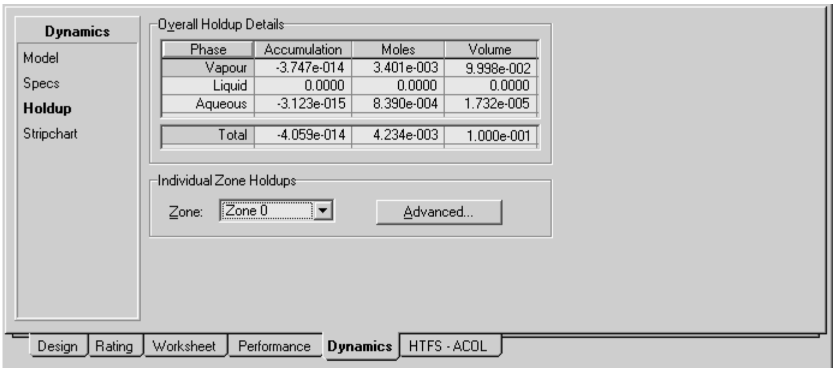
Dynamic Parameters	Description
Fluid Volume	Specify the Air Cooler holdup volume.
Mass Flow	The mass flow of process stream through the Air Cooler is calculated.
Exit Temperature	The exit temperature of the process stream.

Holdup Page

Refer to **Section 1.3.3 - Holdup Page** for more information.

The Holdup page contains information regarding the properties, composition, and amount of the holdup.

Figure 4.9



Refer to **Zone Information** section for more information.

The **Zone** drop-down list enables you to select and view the holdup data for each zone in the operation.

The Air Cooler operation only has one zone.

Stripchart Page

Refer to **Section 1.3.7 - Stripchart Page / Tab** for more information.

The Stripchart page allows you to select and create default strip charts containing various variable associated to the operation.

4.1.8 HTFS - ACOL Tab

For more information about ACOL data input, refer to the **ACOL Reference Guide**.

Also, refer to the ACOL Online Help for information about specific input fields.

This tab allows you to access certain ACOL functions. To access the functions on this tab, you must do the following:

- Install and license ACOL6.4.
- Select HTFS-Engines from the Air Cooler Model drop-down list on the Parameters page of the Design tab.

The HTFS-Engines option runs only in Steady State mode.

If you provide more data than is required, ACOL will perform consistency checks and warn you of any discrepancies.

ACOL Simulation Modes

ACOL has eight different simulation modes, four of which are recognized by HYSYS. Each mode calculates a different variable based on the data you supply. HYSYS checks the data entered for the air-cooler to determine if ACOL can run, then which mode ACOL will run based on the supplied data. HYSYS then sends the data to ACOL.

The following tables list and describe the criteria used by HYSYS to determine the air cooler status messages, whether or not ACOL can run, and which mode ACOL will run.

All simulation modes

The following applies unless specified differently:

Criteria	Value
Air inlet temperature	specified
Air outlet temperature	not specified
Pressure drop	not specified
Process inlet temperature	specified
Process outlet temperature	not specified

ACOL Simulation 9

Calculation of the outlet temperature:

Criteria	Value
Process inlet temperature	specified
Process outlet temperature	not specified

Criteria	Value
Airflow	specified
Process flow rate	specified

ACOL Simulation 1

Calculation of the inlet temperature:

Criteria	Value
Process inlet temperature	not specified
Process outlet temperature	specified
Airflow	specified
Process flow rate	specified
Process inlet temperature	not specified
Process outlet temperature	specified

ACOL Simulation 3

Calculation of the process mass flow rate:

Criteria	Value
Process inlet temperature	specified
Process outlet temperature	specified
Airflow	specified
Process flow rate	not specified

ACOL Simulation 4

Calculation of the air mass flow rate:

Criteria	Value
Process inlet temperature	specified
Process outlet temperature	specified
Airflow	not specified
Process flow rate	specified

Importing and Exporting ACOL Input Files

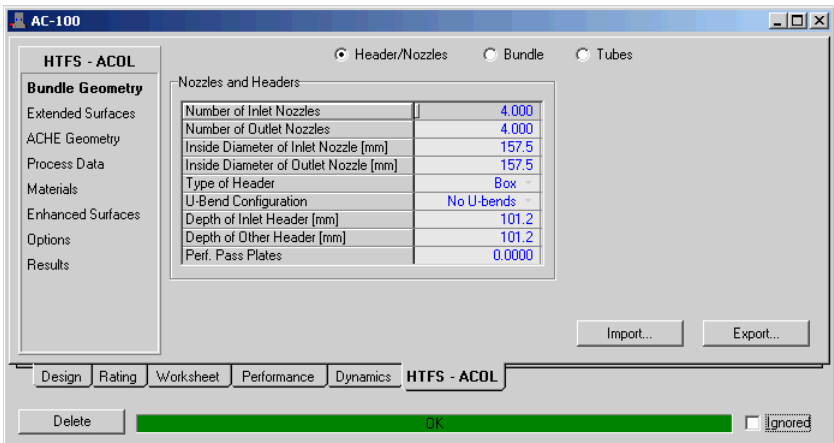
Import and Export buttons appears on every page of the HTFS-ACOL tab. These buttons allow you to import existing ACOL data or export the current data. The file format used is ACOL Input files [***.aci**].

Bundle Geometry Page

The Bundle Geometry page content changes depending on the radio button you select.

Headers/Nozzles Radio Button

Figure 4.10

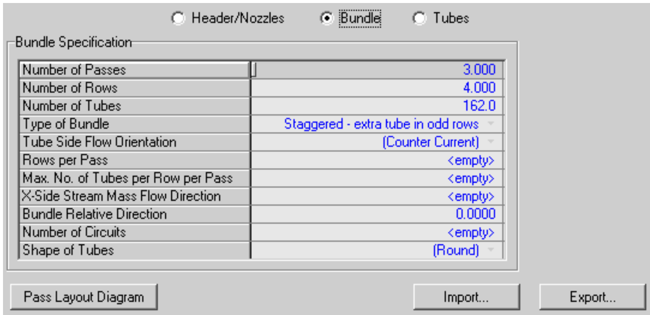


The following table describes the fields that appear when you click the Headers/Nozzles radio button.

Object	Description
Number of Inlet Nozzles	Enter the number of inlet nozzles per bundle. Too few nozzles can cause excessive pressure losses and possibly erosion of the nozzles and headers. Default value is 1.
Number of Outlet Nozzles	Enter the number of outlet nozzles per bundle. If a phase change occurs through the bundle then it may be appropriate to have a different number of nozzles of different size to the inlet nozzles. Default value is 1.
Inside Diameter of Inlet Nozzle	Enter the inside diameter of the inlet nozzles. Defaults to the highest preferred diameter which gives a momentum flux (rV2) less than 6000 kg/m s^2. Preferred sizes are; 50 mm, 100 mm, 150 mm, 200 mm, and so forth.
Inside Diameter of Outlet Nozzle	Enter the inside diameter of the outlet nozzles. Defaults to the highest preferred diameter which gives a momentum flux (rV2) less than 6000 kg/m s^2. Preferred sizes are; 50 mm, 100 mm, 150 mm, 200 mm, and so forth.
Type of Header	Type of Header options: Box, D-header, Plug, Cover Plate, or Manifold.
U-Bend Configuration	U-Bend Configuration options: No-bends, U-bends in alternate passes, or U-bends in every pass.
Depth of Inlet Header	Enter the depth of the header at the tubeside fluid inlet. For a D-header, this will be the maximum depth of the D-section. Default is 300mm (11.8 in) for Air-cooled Heat Exchangers.
Depth of Other Header	Enter the depth of the other header. The other header is at the side opposite to the inlet header. For an odd number of passes, this will be the outlet header. For a D-header, the depth will be the maximum depth of the D-section. Default is 150mm (5.9 in) for Air-cooled Heat Exchangers.
Perf. Pass Plates	Enter the average number of velocity heads lost through each perforated plate in the headers. Perforated pass plates are usually fitted to strengthen the header in high-pressure applications. Default value is 0.0

Bundle Radio Button

Figure 4.11



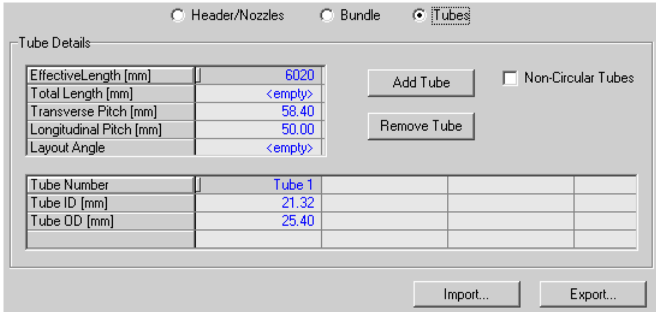
The following table describes the fields that appear when you click the Bundle radio button.

Object	Description
Number of Passes	**Required.** Must be <= 50. With four or more number of passes, the exchanger tends towards the ideal of a pure counter current or co-current exchanger.
Number of Rows	**Required.** Must be <= 100.
Number of Tubes	**Required.** Must be < 1000.
Type of Bundle	There are five types of bundle layouts available from the drop-down list; the bundle layout affects the allowable number of tubes. *NumberOfTubesInARow = NumberOfTubes/ NumberOfRows* If the NumberOfTubesInARow does not have a remainder, then only these bundles can be used: • Inline • Staggered - even rows to the right • Staggered - even rows to the left If the NumberofTubesInARow has a remainder, then only two bundles can be used: • Staggered - extra tubes in odd rows • Staggered - extra tube in even rows
Tube Side Flow Orientation	Select the orientation of the tubeside flow with respect to the X-side flow. This item is used only to correctly set up a symmetrical bundle. It does not apply to a non-symmetrical bundle as the tubeside flow orientation is explicitly set when the bundle is defined using the Pass Layout Window. Select from Counter-current (default), Cross-flow, Co-current

Object	Description
Rows per Pass	Enter the number of tube rows occupied by each tubeside pass. Only to be used when specifying symmetrical bundles. When specifying non-symmetrical bundles use the Pass Layout Window to specify the bundle.
Max. No. Tubes per Row per Pass	Enter the maximum number of tubes in each row occupied by each pass. Only to be used when specifying symmetrical bundles. When specifying non-symmetrical bundles use the interactive bundle specification feature.
X-Side Stream Mass Flow Orientation	Defines the X-side flow orientation relative to the Bundle direction. Enter 0 (vertical-up), 45, 90 (horizontal) or 180 (vertical-down). Default value is 0.
Bundle Relative Direction	Defines the angle of orientation of the bundle relative to the X-side Stream Mass Flow Direction (XSFD) in the range -90° to +90°. If 0° (default) is entered the tubes are always horizontal regardless of the X-side Stream Mass Flow Direction.
Number of Circuits	Enter the number of times a basic pass layout pattern appears in the bundle. The repeat facility is used when a basic pass layout pattern is to be repeated a number of times across the bundle. This feature is most likely to be of use in air-conditioning coils with U tube circuits. It may only be used: a) with inline bundles and staggered bundles with the same number of tubes per row or, b) when X-side stream inlet conditions do not vary across the bundle. When using the repeat facility, count the original section as 1 (Default).
Shape of Tubes	Select from Round (default), Oval, or Flat. If Oval or Flat tubes are selected, the geometric data for the tube should be entered for each tube type on the Non-circular Tubes page (click the Tubes radio button).The geometric data for each fin type can be entered on the Extended Surfaces page.
Pass Layout Diagram button	Displays a Pass Layout diagram that allows you to specify the pass arrangement according to your requirements.

Tubes Radio Button

Figure 4.12



The following table lists and describes the fields that appear when you click the Tubes radio button.

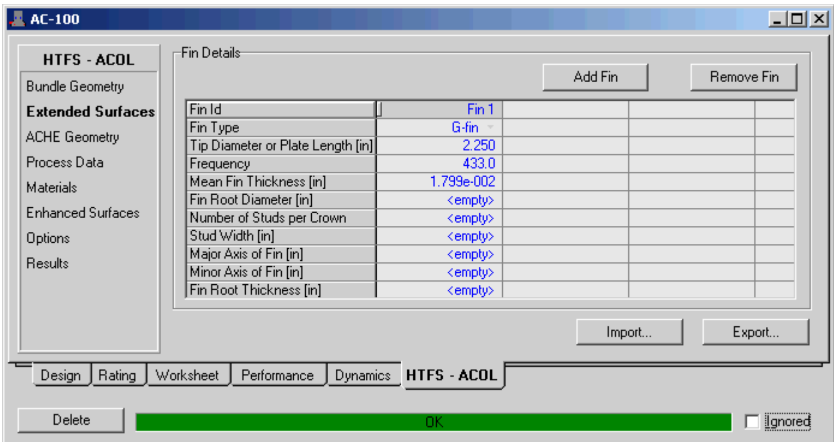
Object	Description
Common Options	
Add Tube button	Adds a tube to the air cooler.
Remove Tube button	Removes a tube from the air cooler.
Effective Length	This is the length of tube that is transferring heat. Inactive parts of a tube are where it fits into the tubesheets and comes into contact with tube supports. Include these parts in the Total Tube Length. Default is 6000mm (19.7 ft) for Air-cooled Heat Exchangers.
Total Length	This is the total length of tube including the ends fitted into the tubesheets and where the tube comes into contact with tube supports.This is used for tubeside pressure drop calculations only. Default value is the Effective tube length.
Transverse Pitch	This is the distance between the centre-lines of consecutive tubes in the same tube row. Default is 2.3 times Tube OD for Air-cooled Heat Exchangers

Object	Description
Longitudinal Pitch	If you have a standard TEMA tube layout, for example triangular (30°), rotated square (45°), rotated triangular (60°), or square (90°), then use the layout angle. If you have a non-standard tube layout then use this item. The plain tube correlations are only valid for the standard TEMA tube layout given above so use layout angle in this case. For uncommonly large longitudinal pitches, you may have to allow for a reduction in the heat transfer coefficient separately from ACOL. Currently ACOL does not allow for this effect. There is no default value. The value will be calculated from the Transverse Pitch and the Layout Angle.
Layout Angle	Use this field to enter the layout angle for a standard TEMA tube layout. • 30° – triangular arrangement (default) • 45° – rotated square arrangement • 60° – rotated triangular arrangement • 90° – square arrangement (for in-line banks only) If you have a non-standard tube layout, in other words one, which would give a layout angle not in the above list then, input longitudinal pitch instead of this item. Use this item for plain tubes, as the correlations are only valid for the standard TEMA tube layouts. Default Value is 30°.
Tube Details group options	
Tube Number	Displays the system defined tube number. If you have more than one tube type defined, then corresponding input cells appear on the Extended Surfaces and Materials pages.
Tube ID	Up to 4 Tube Diameters may be specified. Default values for Tube ID(1): Tube ID(1) = Tube OD(1) – 3.3mm (0.13in) for Air-cooled Heat Exchangers. Other tube types default to Tube ID(1).
Tube OD	Up to 4 Tube Diameters may be specified. API661 recommends 25.4 mm or 1 inch as the minimum outside diameter.
Non-Circular Tube Details group options	
Non-Circular Tubes checkbox	Select this checkbox to specify non-circular tube parameters. When you click this checkbox, the fields listed below appear.
Tube Number	Displays the system defined tube number.
Major Axis on Outside of Tube	Allows you to specify the length of the flatter side of the tube.

Object	Description
Minor Axis on Outside of Tube	Allows you to specify the length of 'short' side of the tube.
Tube Wall Thickness	Allows you to specify the tube wall thickness.

Extended Surfaces Page

Figure 4.13



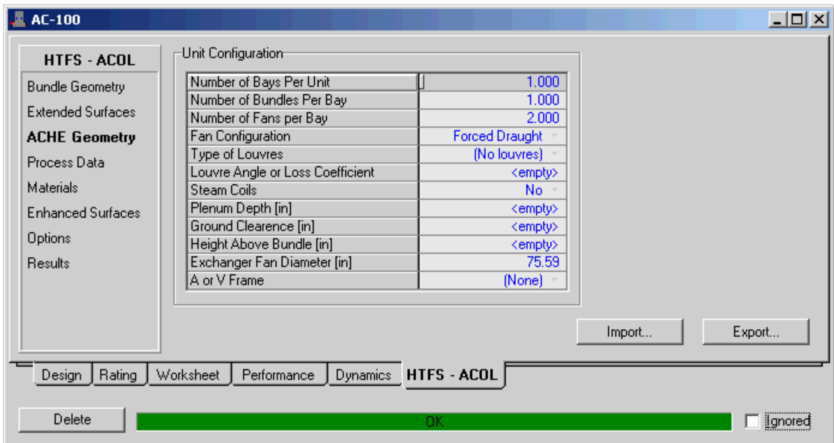
The following table lists and describes the objects on this page.

Object	Description
Add Fin	Click this button to add a fin. The fin parameter set appears in the Fin Details table. If you have finned tubes, you will need to supply tube and fin details on the Bundle Geometry and Extended Surfaces pages.
Remove Fin	Click this button to remove the select fin parameter set.
Fin ID	Displays the system generated Fin number.
Fin Type	Allows you to select a fin type from a drop-down list: • Integral • G-fin (embedded) (default) • Modified G-fin • L-finned • Bi-metallic or extruded • Shoulder-grooved • Tube-in-plate • Plain tubes • Serrated fins • Low fins • Circular studs • Rectangular studs • Elliptical studs • Lenticular studs • Chamfered studs

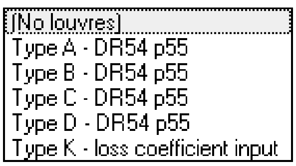
Object	Description
Tip Diameter or Plate Length	For a finned or studded tube, enter the fin (or stud) tip diameter. Default is 2.25 times Tube OD for Air-cooled Heat Exchangers. For tube-in-plate fins, enter the plate length in the direction of the X-side flow (from the leading edge to the trailing edge of the plate). This will be calculated if left blank.
Frequency	This is the number of fins per unit length or the number of stud crowns per unit length. Default is 433 fins/m (11 fins/inch) for Air-cooled Heat Exchangers.
Mean Fin Thickness	For fins made by wrapping ribbon around the base tube, the fin thickness is usually thinner than the ribbon thickness. Default is 0.28mm (0.011in) for Air-cooled Heat Exchangers
Fin Root Diameter	Enter the root diameter for Integral, L-finned, Extruded tubes or Shoulder-grooved fins. For other fin types, the fin root diameter is the base tube outside diameter. The Common Fin Root Diameter applies to the whole bundle unless a local values is used. Defaults to the tube outside diameter.
Number of Studs per Crown	This is the number of studs making up a crown.
Stud Width	This item is not required for circular studs.
Major Axis of Fin	This is the length of the ‘long’ side of the tube. Default is 54 mm (2.13 in).
Minor Axis of Fin	This is the length of the ‘short’ side of the tube. Default value is 34 mm (1.34 in).
Fin Root Thickness	For L shape or bimetallic fins. Fin Root Thickness is used in place of Fin Root Diameter for round fins. Default value is 0.0.

ACHE Geometry Page

Figure 4.14



The following table lists and describes some of the objects on the ACHE Geometry page.



Object	Description
Number of Bays per Unit	**Required**. Range 1-99. Default is 1.
Number of Bundles per Bay	**Required**. Range 1-12. Default is 1.
Number of Fans per Bay	**Required**. Range 1-6. Default is 2.
Fan Configuration	Select from Forced Draught, Induced Draught, or No fans.
Type of Louvres	Select the type of louvres required for the air cooler. Options appear in the image to the left.
Louvre Angle or Loss Coefficient	Enter either the louvre opening angle (for louvre types A-D) or the loss coefficient (for louvre type K). An angle of 0° is fully open and 90° is fully closed.
Steam Coils	Select Yes or No (default) depending on whether a steam coil is fitted. This item is used only in the calculation of the X-side pressure drop. Steam coils are assumed to consist of one row of tubes with the same tube geometry as the first type of fin but with twice the transverse pitch.
Plenum Depth	This is distance from the bundle side of the fan ring to the bundle. Defaults to 0.4 times the exchanger fan diameter.

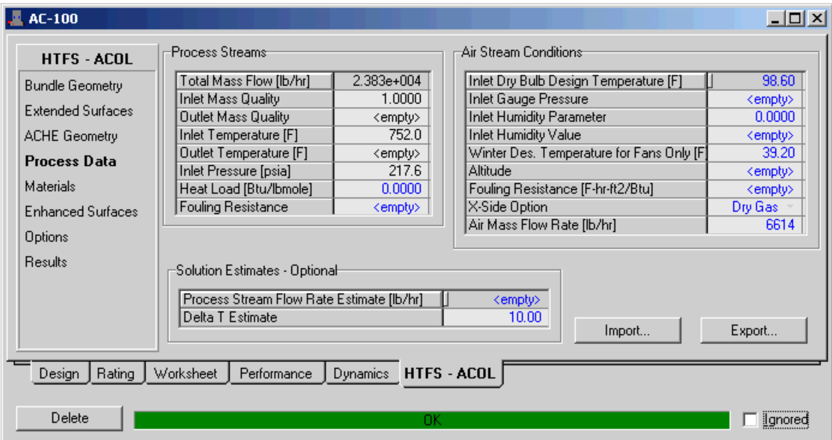
Object	Description
Ground Clearance	This is the distance from the ground to the fan inlet for a forced draught exchanger or to the bundle entry for an induced draught exchanger. Defaults to1.5 times the exchanger fan diameter.
Height Above Bundle	This is the distance from the top of the bundle to the exchanger exit. Use only with the Natural Convection simulation option. The hardware height acts as a 'chimney' filled with hot air. For forced draught exchangers this will typically be the height of a wind skirt above the bundle. For induced draught exchangers it will be the distance to the top of the fan casing. Default value is 0.0.
Exchanger Fan Diameter	This is used to calculate fan related pressure losses and fan noise levels. The fan diameter cannot be larger than the bay width. Default calculated to give 40% bundle coverage per fan.
A or V Frame	The default is (None).

Process Data Page

Refer to **ACOL Simulation Modes** section for more information.

HYSYS uses information in the first six fields to determine if ACOL can run, and what mode it will use.

Figure 4.15



You cannot edit the values in black text. These values are determined using input on other tabs in the Air Cooler property view.

The following table lists and describes the objects on this page.

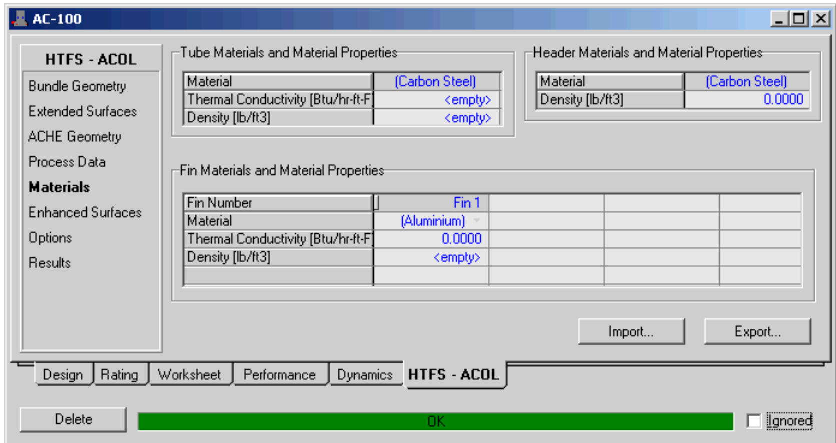
Object	Description
Process Steams group	
Total Mass Flow	Displays the process stream mass flow calculated by HYSYS.
Inlet Mass Quality	Displays the default value.
Outlet Mass Quality	Displays the system defined outlet mass quality.
Inlet Temperature	Displays the stream inlet temperature as defined on the Worksheet tab.
Outlet Temperature	Displays the stream outlet temperature, if available.
Inlet Pressure	Displays the stream inlet pressure as defined on the Worksheet tab.
Heat Load	You may enter the heat load directly, or omit it and leave ACOL to calculate it from the stream flowrate and inlet and outlet conditions. ACOL will use the input heat load to calculate the duty ratio (heat load calculated/heat load input), otherwise it will use the input tubeside stream conditions.
Fouling Resistance	Allows you to specify the fouling resistance of the process stream.
Air Stream Conditions group	
Inlet Dry Bulb Design Temperature	This is the temperature of the incoming air; it has a significant effect on the overall heat transfer area required. This is a useful parameter for helping to determine Annual Fan Power Consumption.
Inlet Gauge Pressure	This is the gauge pressure of the air at entry to the bundle. This item is intended primarily for ducted systems where there may be a slight positive air inlet pressure. Negative values may also be used.The default air pressure is the International Standard Atmosphere at sea level, 1013mbar. Use either or both inlet gauge pressure and altitude to specify the actual inlet air pressure.
Inlet Humidity Parameter	Select the way in which the Inlet Humidity Value will be expressed: • Humidity ratio (default) • Relative humidity The only two-phase system that ACOL can handle on the X-side is the condensation of water vapour from a humid air stream. Important note: If you want to use this parameter, ensure that you have selected Humid Air for the X-side Option
Inlet Humidity Value	This is the value of the air inlet humidity in the way selected by the Inlet Humidity Parameter.

Object	Description
Winter Des. Temperature for Fans Only	This is the value for the X-side Stream Winter Inlet Temperature (or Minimum Ambient Temperature) and is used for calculating maximum fan power consumption only. Only relevant to forced draught exchangers. Default value is 0°C (32°F).
Altitude	This is the height of the unit above sea level. You can use either or both inlet gauge pressure and altitude to specify the actual inlet air pressure.The default air pressure is the International Standard Atmosphere at sea level, 1013mbar.
Fouling Resistance	Allows you to specify the fouling resistance of the air stream.
X-Side Option	This is the fluid you wish to use on the X-side. Select from Dry Air (default), Humid Air, or Dry Gas. Dry Air is appropriate for air-cooled heat exchangers and other heat exchangers where air is being heated. Dry Gas is appropriate for waste heat recovery units where gases such as flue gases are being cooled. Also for any exchanger where gases other than or including air are handled. ACOL cannot handle condensation of any of the components of the gas stream.
Air Mass Flow Rate	Allows you to specify the air mass flow rate for the air stream. You can also edit this value on the Rating tab.
Solution Estimates - Optional group For ACOL to run, it must have initial values for this group. Only the estimate that applies to the current calculation will be used.	
Process Stream Flow Rate Estimate	Provides an initial value for ACOL calculations. If you do not enter a value, the following estimates are used. When calculating the process mass flow rate: *Process Mass Flow = 5kg/s* When calculating the air mass flow rate: *No estimate required*
Delta T Estimate	Provides an initial value for ACOL calculations. If you do not enter a value, the following estimates are used. When calculating the outlet process temperature: *Outlet Process temp = Inlet Process temp - 10°C* When calculating the inlet process temperature: *Inlet Process temp = Outlet Process temp + 10°C*

Materials Page

On this page you can define the tube, header and fin materials and material properties. The default material for tubes and headers is carbon steel; the default for fins is aluminium.

Figure 4.16

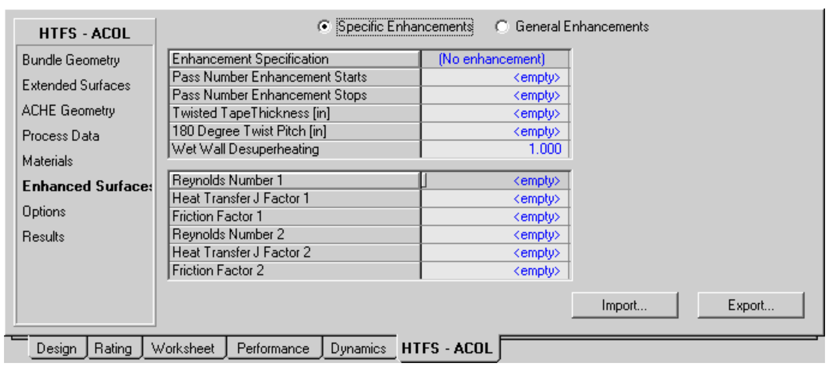


Enhanced Surfaces Page

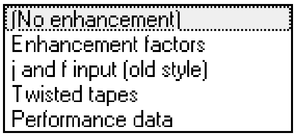
This page changes depending on which radio button you select.

- Specific Enhancements Radio Button

Figure 4.17



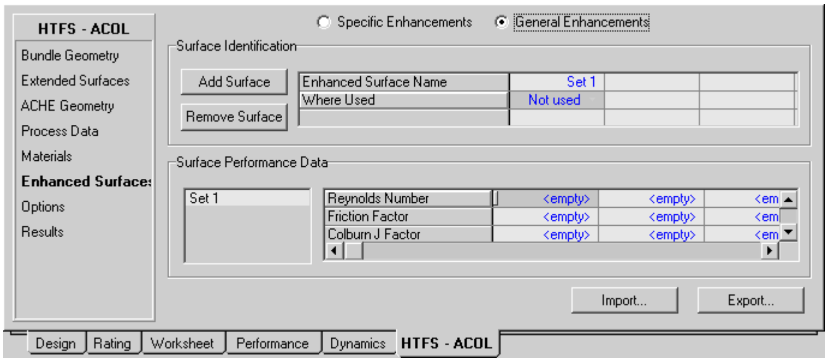
The following table lists and describes some of the objects available for the Specific Enhancements option.



Object	Description
Enhancement Specification	Select the type of enhancement specification from the drop-down list. Available options appear in the image to the left.
Pass Number Enhancement Starts	Enter the pass number from which (and including) the tube enhancement is to take effect. This allows you to specify tubeside enhancement where it might be most effective.
Pass Number Enhancement Stops	Enter the pass number at which (and including) the tube enhancement is to stop. If this item is left blank and tubeside enhancement has been specified, then the enhancement will stop at the last pass.
Twisted Tape Thickness	This is the thickness of the twisted tape insert. Default value is 0.5 mm (0.02 in).
180 Degree Twist Pitch	This is the pitch of the twisted tape insert as it completes one 180-degree twist. Default value is 50 mm (2 in).
Wet Wall Desuperheating	Select YES for wet wall (or NO for dry wall) desuperheating. Wet wall desuperheating occurs when the bulk temperature of a stream is above the dew point, but the local wall temperature is below the dew point. If the wet wall calculation is selected, the program corrects the heat transfer rate in the desuperheating zone to allow for condensation occurring at the wall. When the alternative dry wall calculation is selected the program uses the single phase gas coefficient until the bulk vapour temperature reaches the dew point. As a rule, dry wall coefficients are usually lower than wet wall coefficients, and more conservative. Default is Yes
Reynolds Number	This field allows you to enter values of Reynolds Number for the first and second points which correspond with input values of tubeside heat transfer j factors and friction factors. The reference diameter is the tube inside diameter. A log-log interpolation is performed between two points. Extrapolation is not permitted.
Heat Transfer J Factor	This field allows you to enter values of the heat transfer j factor corresponding to the values of the Reynolds Number for Points 1 and 2. This is particularly useful for specifying the performance of tube inserts. A log-log interpolation is performed between two points. Extrapolation is not permitted.
Friction Factor	This field allows you to enter values of the friction factor corresponding to the values of the Reynolds Number for Points 1 and 2. This is particularly useful for specifying the performance of tube inserts. A log-log interpolation is performed between two points. Extrapolation is not permitted.

- General Enhancements Radio Button

Figure 4.18



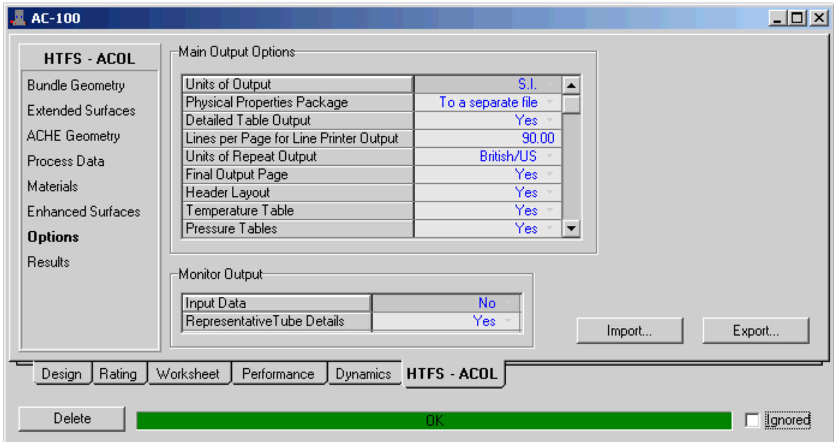
The following table lists and describes some of the objects available for the General Enhancements option.

Object	Description
Surface Identification group	
Add Surface button	Adds another surface set to the matrix.
Remove Surface button	Removes a surface set from the matrix.
Enhanced Surface Name	Displays the system generated set name. Maximum number of surfaces = 20.
Where Used	Defines where the enhanced surface is used.
Surface Performance Data group	
Set list	Displays the list of available sets.
Reynolds Number	Allows you to specify the Reynolds Number for the selected set. You can enter up to four values.
Friction Factor	Allows you to specify the friction factor for the selected set. You can enter up to four values.
Colburn J Factor	Allows you to specify the Colburn J Factor for the selected set. You can enter up to four values.

Options Page

This page determines what appears on the Results page.

Figure 4.19



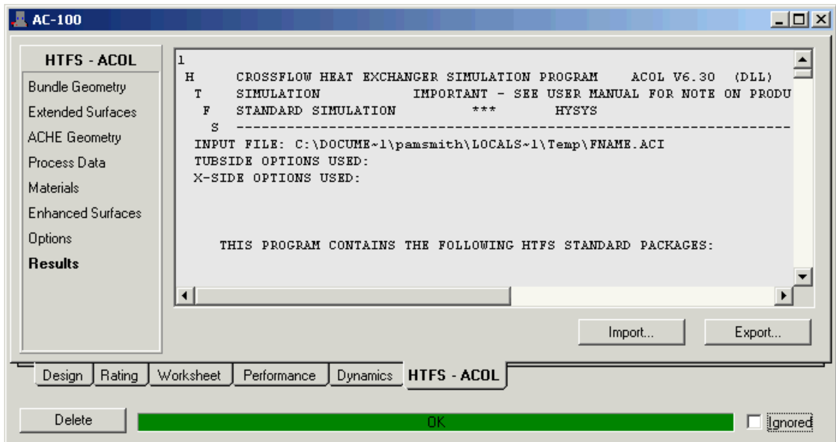
Object	Description
Main Output Options group	
Units of Output	Determines the output data units. Choose from S.I., British/US, and Metric.
Physical Properties Package	Determines where the output data goes: line printer, separate file, or no output
Detailed Table Output	Determines the output table format.
Lines per Page for Line Printer Output	Sets the number of lines on a page for printed output.
Units of Repeat Output	Sets the units for repeated output on the Results page; contains the same options as Units of Output.
Final Output Page	Select Yes if you require the final output page in the lineprinter output.
Header Output	Select Yes to show headers in the lineprinter output.
Temperature Table	Select Yes to show the temperature table in the lineprinter output.
Pressure Tables	Select Yes to show the pressure tables in the lineprinter output.
Monitor Output group	
Input Data	Use the default setting.
Representative Tube Details	Use the default setting.

The Monitor Output group is used for debugging purposes only. Use the default settings.

Results Page

This page displays the result of the ACOL calculations. Set the format of the Results page on the Options page.

Figure 4.20



4.2 Cooler/Heater

The Cooler and Heater operations are one-sided heat exchangers. The inlet stream is cooled (or heated) to the required outlet conditions, and the energy stream absorbs (or provides) the enthalpy difference between the two streams. These operations are useful when you are interested only in how much energy is required to cool or heat a process stream with a utility, but you are not interested in the conditions of the utility itself.

The difference between the Cooler and Heater is the energy balance sign convention.

4.2.1 Theory

The Cooler and Heater use the same basic equation.

Steady State

The primary difference between a cooler and a heater is the sign convention. You specify the absolute energy flow of the utility stream, and HYSYS then applies that value as follows:

- For a Cooler, the enthalpy or heat flow of the energy stream is subtracted from that of the inlet stream:

$$Heat\ Flow_{\text{inlet}} - Duty_{\text{cooler}} = Heat\ Flow_{\text{outlet}} \tag{4.12}$$

- For a Heater, the heat flow of the energy stream is added:

$$Heat\ Flow_{\text{inlet}} + Duty_{\text{cooler}} = Heat\ Flow_{\text{outlet}} \tag{4.13}$$

Dynamic

The Cooler duty is subtracted from the process holdup while the Heater duty is added to the process holdup.

For a Cooler, the enthalpy or heat flow of the energy stream is removed from the Cooler process side holdup:

$$M(H_{in} - H_{out}) - Q_{cooler} = \rho\frac{d(VH_{out})}{dt} \tag{4.14}$$

For a Heater, the enthalpy or heat flow of the energy stream is added to the Heater process side holdup:

$$M(H_{in} - H_{out}) + Q_{heater} = \rho\frac{d(VH_{out})}{dt} \tag{4.15}$$

where:

M = process fluid flow rate

ρ = density

H = enthalpy

Q_{cooler} = *cooler duty*

Q_{heater} = *heater duty*

V = volume shell or tube holdup

Pressure Drop

The pressure drop of the Cooler/Heater can be determined in one of two ways:

- Specify the pressure drop manually.
- Define a pressure flow relation in the Cooler or Heater by specifying a k-value.

If the pressure flow option is chosen for pressure drop determination in the Cooler or Heater, a k value is used to relate the frictional pressure loss and flow through the Cooler/Heater.

The relation is similar to the general valve equation:

$$flow = \sqrt{density} \times k\sqrt{P_1 - P_2} \tag{4.16}$$

This general flow equation uses the pressure drop across the heat exchanger without any static head contributions. The quantity, P_1 - P_2, is defined as the frictional pressure loss which is used to “size” the Cooler or Heater with a k-value.

Dynamic Specifications

In general, two specifications are required by HYSYS in order for the Cooler/Heater unit operation to fully solve in Dynamic mode:

Dynamic Specifications	Description
Duty Calculation	The duty applied to the Cooler/Heater can be calculated using one of three different models: • Supplied Duty • Product Temp Spec • Duty Fluid Specify the duty model in the Model Details group on the Specs page of the Dynamics tab.
Pressure Drop	Either specify an Overall Delta P or an Overall K-value. Specify the Pressure Drop calculation in the Dynamic Specifications group on the Specs page of the Dynamics tab.

4.2.2 Heater or Cooler Propety View

There are two ways that you can add a Heater or Cooler to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.

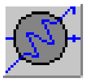
2. Click the **Heat Transfer Equipment** radio button.

3. From the list of available unit operations, select **Cooler** or **Heater**.
4. Click the **Add** button.

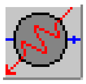
OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.
2. Double-click the **Cooler** icon or the **Heater** icon.



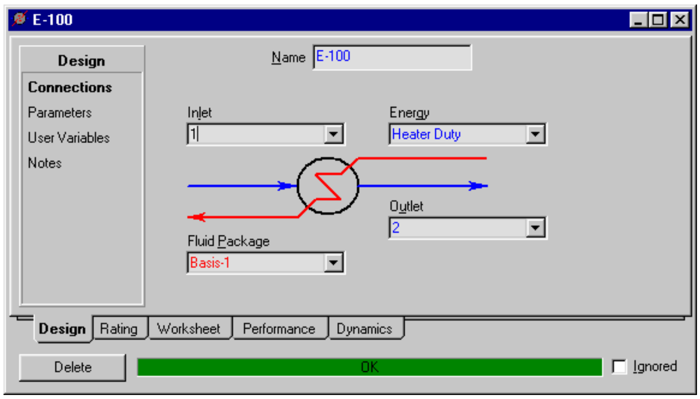
Cooler icon



Heater icon

The Cooler or Heater property view appears.

Figure 4.21



4.2.3 Design Tab

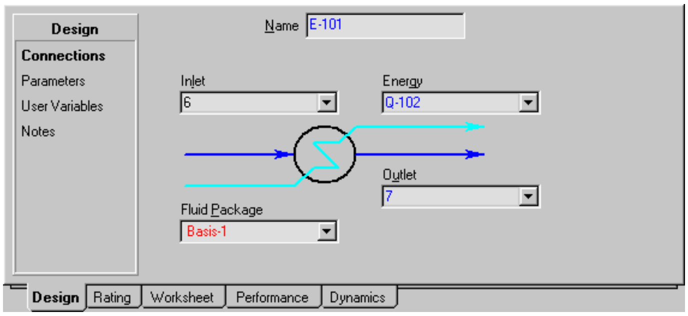
The Design tab contains the following pages:

- Connections
- Parameters
- User Variables
- Notes

Connections Page

The Connections page is used to define all of the connections to the Cooler/Heater. You can specify the inlet, outlet, and energy streams attached to the operation on this page. The name of the operation can be changed in the Name field.

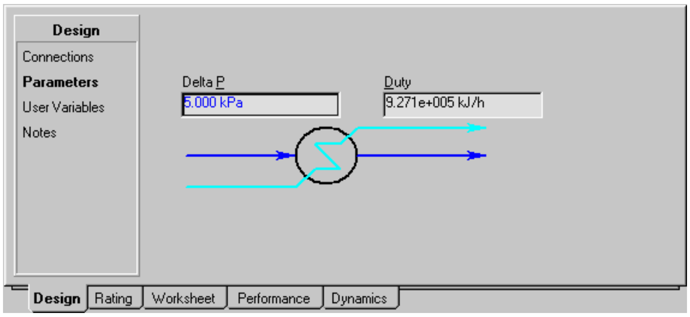
Figure 4.22



Parameters Page

The applicable parameters are the pressure drop (Delta P) across the process side, and the duty of the energy stream. Both the pressure drop and energy flow can be specified directly or can be determined from the attached streams.

Figure 4.23



HYSYS uses the proper sign convention for the unit you have chosen, so you can enter a positive duty value for both heater and cooler.

You can specify a negative duty value, however, be aware of the following:

- For a Cooler, a negative duty means that the unit is heating the inlet stream.
- For a Heater, a negative duty means that the unit is cooling the inlet stream.

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor that allows you to record any comments or information regarding the specific unit operation, or the simulation case in general.

4.2.4 Rating Tab

You must specify the rating information only when working with a dynamics simulation.

Nozzles Page

Refer to **Section 1.3.6 - Nozzles Page** for more information.

On the Nozzles page, you can specify nozzle parameters on both the inlet and outlet streams connected to a Cooler or Heater**.** The addition of nozzles to Coolers and Heaters is relevant when creating dynamic simulations.

Heat Loss Page

Rating information regarding heat loss is relevant only in Dynamic mode. The Heat Loss page contains heat loss parameters that characterize the amount of heat lost across the vessel wall.

In the Heat Loss Model group, you can choose either a Simple or Detailed heat loss model or no heat loss through the vessel walls.

Simple Model

The Simple model allows you to either specify the heat loss directly, or have the heat loss calculated from the specified values:

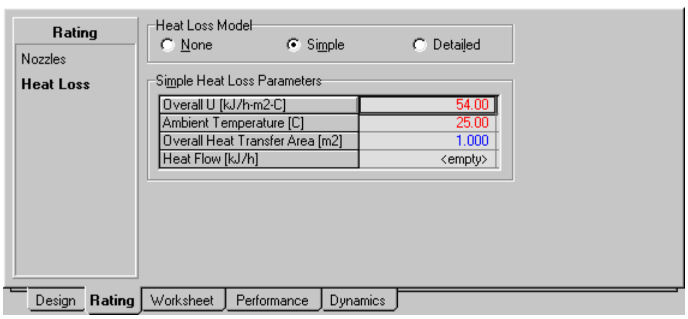
- Overall U value
- Ambient Temperature

The heat transfer area, A, and the fluid temperature, T_f, are calculated by HYSYS using the following equation:

$$Q = UA(T_f - T_{amb}) \tag{4.17}$$

For a Cooler, the parameters available for the Simple model appear in the figure below.

Figure 4.24



The simple heat loss parameters are as follows:

- Overall Heat Transfer Coefficient
- Ambient Temperature
- Overall Heat Transfer Area
- Heat Flow

The heat flow is calculated as follows:

$$Heat\ Flow = UA(T_{Amb} - T) \tag{4.18}$$

where:

U = *overall heat transfer coefficient*

A = *heat transfer area*

T_{Amb} = *ambient temperature*

T = *holdup temperature*

Heat flow is defined as the heat flowing into the vessel. The heat transfer area is calculated from the vessel geometry. The ambient temperature, T_{Amb}, and overall heat transfer coefficient, U, can be modified from their default values shown in red.

Detailed Model

Refer to **Section 1.6.1 - Detailed Heat Model** in the **HYSYS Dynamic Modeling** guide for more information.

The Detailed model allows you to specify more detailed heat transfer parameters.

The HYSYS Dynamics license is required to use the Detailed Heat Loss model.

4.2.5 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the unit operation.

The PF Specs page is relevant to dynamics cases only.

4.2.6 Performance Tab

The Performance tab contains pages that display calculated stream information. By default, the performance parameters include the following stream properties:

- Pressure
- Temperature
- Vapour Fraction
- Enthalpy

Other stream properties can be viewed by adding them to the Viewing Variables group on the Setup page.

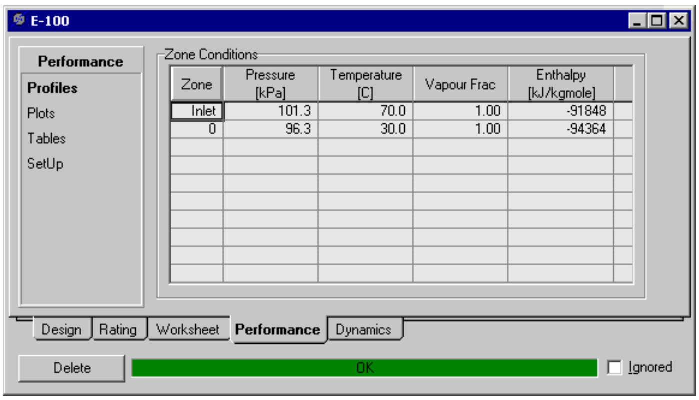
All information appearing on the Performance tab is read-only. The Performance tab contains the following pages:

- Profiles
- Plots
- Tables
- Setup

Profiles Page

In Steady State mode, HYSYS calculates the zone conditions for the inlet zone only, regardless of the number of zones specified.

Figure 4.25



Plots Page

On the Plots page, you can graph any of the default performance parameters to view changes that occur across the operation.

In Steady State mode, stream property readings are taken only from the inlet and outlet streams for the plots. As such, the resulting graph is always a straight line. The property values are not calculated incrementally through the operation.

All default performance parameters are listed in the X Variable and Y Variable drop-down lists below the graph. Select the axis and variables you want to compare, and the plot is displayed.

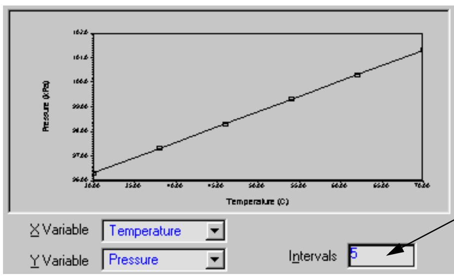
To graph other variables, you need to go to the Setup page and add them to the Selected Viewing Variables group from the Available Variables listed.

Refer to **Section 1.3.1 - Graph Control Property View** for more information.

You can right-click on the graph area to access the graph controls and manipulate the graph appearance.

A temperature - pressure graph for a Cooler, with 5 specified intervals is displayed in the figure below.

Figure 4.26

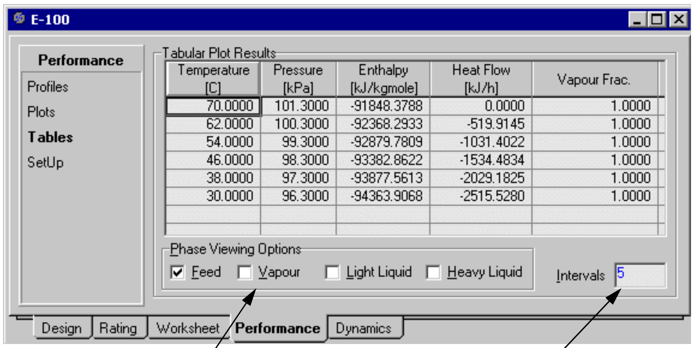


You can specify the number of calculation intervals you want calculated across the graph in the Interval field. This divides the plot line into equally spaced intervals with the values displayed as described on the Tables page.

Tables Page

The Tables page displays the results of the Cooler/Heater in a tabular format. All default values for the pressure, temperature, vapour fraction, and enthalpy calculated for each interval are listed here.

Figure 4.27



You can select what phase options to view by clicking on the checkbox. For some options you need to add variables via the Setup page.

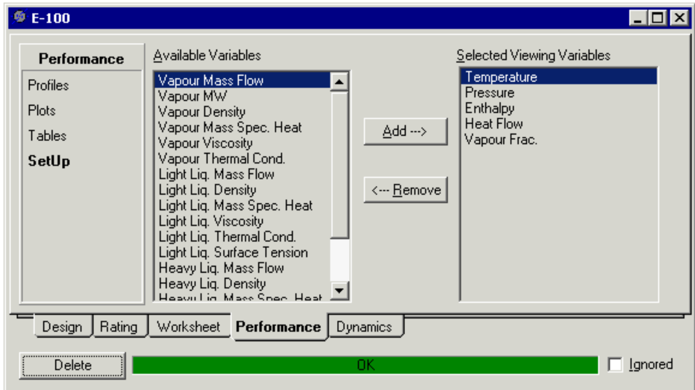
You can specify the number of calculation intervals you want calculated across the data in the Interval field. This divides the data up into equally spaced intervals.

Information on the Tables page is read-only, except the Intervals value.

Setup Page

The Setup page allows you to filter and add variables to be viewed on the Plots and Tables pages.

Figure 4.28



The variables that are listed in the Selected Viewing Variables group are available in the X and Y drop down list for plotting on the Plots page. The variables are also available for tabular plot results on the Tables page based on the Phase Viewing Options selected.

4.2.7 Dynamics Tab

If you are working exclusively in Steady State mode, you do not need to change any of the values on the pages accessible on the Dynamics tab.

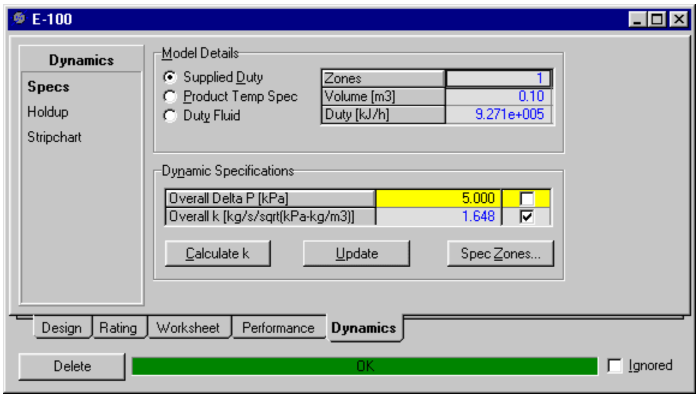
In the Dynamic mode, the values you enter in the Dynamics tab affects the calculation. The Dynamics tab contains the following pages:

- Specs
- Duty Fluid
- Holdup
- Stripchart

Specs Page

The Specs page contains information regarding the calculation of pressure drop across the Cooler or Heater:

Figure 4.29



Zone Information

HYSYS has the ability to partition heat transfer opera6tions into discrete sections called zones. By dividing the unit operation into zones, you can make different heat transfer specifications for individual zones, and therefore more accurately model the physical process.

Specifying the Cooler/Heater with one zone provides optimal speed conditions, and is usually sufficient in modeling accurate exit stream conditions.

Model Details

The Model Details group must be completed before the simulation case solves. The number of zones and the volume of a Cooler/Heater can be specified in this group.

HYSYS can calculate the duty applied to the holdup fluid using one of the three different methods described in the table below.

Model	Description
Supplied Duty	If you select the Supplied Duty radio button, you must specify the duty applied to the Cooler/Heater. It is recommended that the duty supplied to the unit operation be calculated from a PID Controller or a Spreadsheet operation that can account for zero flow conditions.
Product Temp Spec	If you select the Product Temp Spec radio button, you must specify the desired exit temperature. HYSYS back calculates the required duty to achieve the specified desired temperature. This method does not run as fast as the Supplied Duty model.
Duty Fluid	If you select the Duty Fluid radio button, you can model a simple utility fluid to heat or cool your process stream. The following parameters must be specified for the utility fluid on the Duty Fluid page of the Dynamics tab: • Mass Flow • Holdup Mass • Mass Cp • Inlet temperature • Average UA

Dynamic Specifications

The Dynamic Specifications group allows you to specify how the pressure drop is calculated across the Cooler or Heater unit operation. The table below describes the specifications.

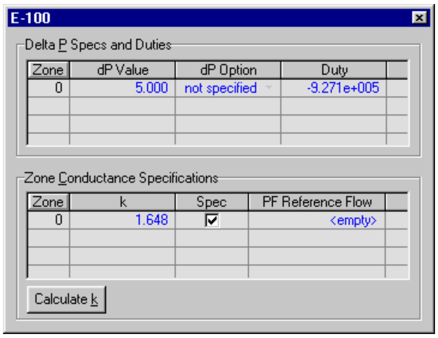
Specification	Description
Overall Delta P	A set pressure drop is assumed across the Cooler or Heater operation with this specification. The flow and the pressure of either the inlet or exit stream must be specified, or calculated from other unit operations in the flowsheet. The flow through the valve is not dependent on the pressure drop across the Cooler or Heater. To use the overall delta P as a dynamic specification, select the corresponding checkbox in the Dynamic Specifications group
Overall k Value	The k-value defines the relationship between the flow through Cooler or Heater and the pressure of the surrounding streams. You can either specify the k-value, or have it calculated from the stream conditions surrounding the unit operation. You can “size” the Cooler or Heater with a k-value by clicking the Calculate k button. Ensure that there is a non zero pressure drop across the Cooler or Heater before the Calculate k button is clicked. To use the k-value as a dynamic specification, select the corresponding checkbox in the Dynamic Specifications group.

The Cooler or Heater unit operation, like other dynamic unit operations, should use the k-value specification option as much as possible to simulate actual pressure flow relations in the plant.

Zone Dynamic Specifications

If the Cooler or Heater operation is specified with multiple zones, you can click the Spec Zones button to define dynamic specifications for each zone.

Figure 4.30



In the Delta P Specs and Duties group, you can specify the following parameters:

Dynamic Specification	Description
dP Value	Allows you to specify the fixed pressure drop value.
dP Option	Allows you to either specify or calculate the pressure drop across the Cooler or Heater. Specify the dP Option with one of the following options: • **user specified**. The pressure drop across the zone is specified by you in the dP Value field. • **non specified**. Pressure drop across the zone is calculated from a pressure flow relationship. You must specify a k-value, and activate the specification for the zone in the Zone Conductance Specifications group.
Duty	A fixed duty can be specified across each zone in the Cooler or Heater unit operation.

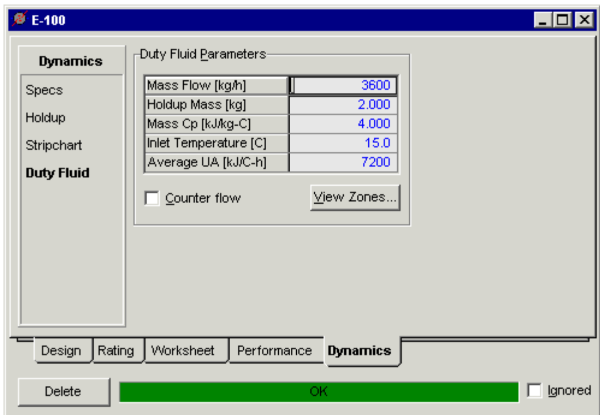
In the Zone Conductance Specifications group, you can specify the following parameters:

Dynamic Specification	Description
k	The k-value for individual zones can be specified in this field. You can either specify the k-value, or have it calculated by clicking the Calculate k button
Specification	Activate the specification if the k-value is to be used to calculate pressure across the zone.

Duty Fluid Page

The Duty Fluid page becomes visible if the Duty Fluid radio button is selected on the Specs page.

Figure 4.31



The Duty Fluid page allows you to enter the following parameters to define your duty fluid:

- Mass Flow
- Holdup mass
- Mass Cp
- Inlet Temperature
- Average UA

The Counter Flow checkbox allows you to specify the direction of flow for the duty fluid. When the checkbox is active, you are using a counter flow.

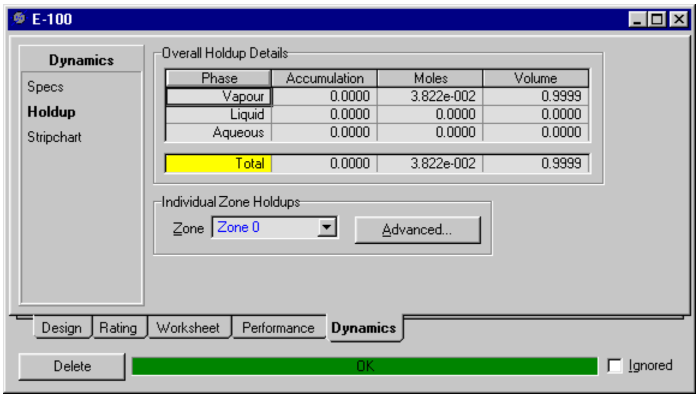
The **View Zones** button displays the duty fluid parameters for each of the zones specified on the Specs page.

Holdup Page

Refer to **Section 1.3.3 - Holdup Page** for more information.

The Holdup page contains information regarding the Cooler or Heater holdup properties, composition, and amount.

Figure 4.32



The Individual Zone Holdups group contains detailed holdup properties for each holdup in the Cooler or Heater. In order to view the advanced properties for individual holdups, you must first choose the individual zone in the **Zone** drop-down list.

Stripchart Page

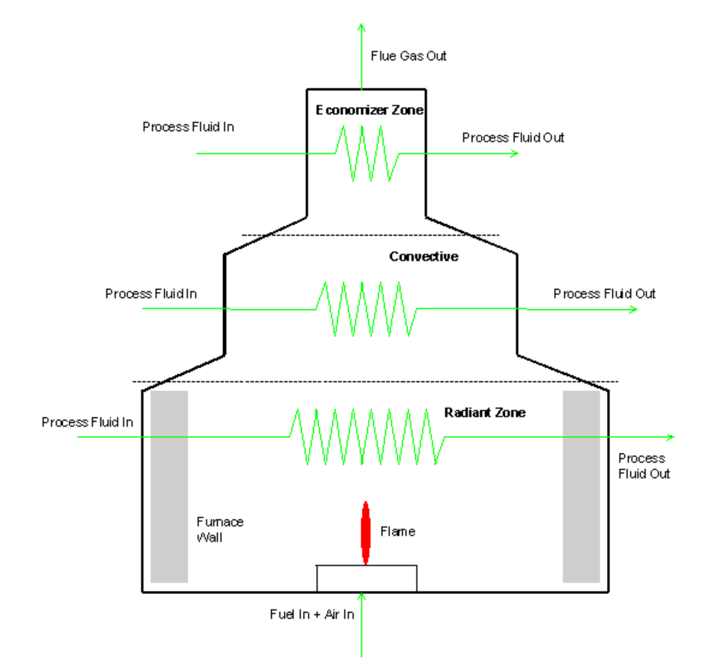
Refer to **Section 1.3.7 - Stripchart Page/Tab** for more information.

The Stripchart page allows you to select and create default strip charts containing various variable associated to the operation.

4.3 Fired Heater (Furnace)

The dynamic Fired Heater (Furnace) operation performs energy and material balances to model a direct Fired Heater type furnace. This type of equipment requires a large amount of heat input. Heat is generated by the combustion of fuel and transferred to process streams. A simplified schematic of a direct Fired Heater is illustrated in the figure below.

Figure 4.33



The Fired Heater operation is available as a dynamic unit operation only.

In general, a Fired Heater can be divided into three zones:

- Radiant zone
- Convective zone
- Economizer zone

To define the number of zones required by the Fired Heater, enter the number in #External Passes field on Connections page of the Design tab.

The Fired Heater operation allows multiple stream connections at tube side in each zone and optional economizer, and convection zone selections. The operation incorporates a single burner model, and a single feed inlet and outlet on the flue gas side.

The following are some of the major features of the dynamic Fired Heater operation:

- Flexible connection of process fluid associated in each Fired Heater zone. For example, radiant zone, convective zone, or economizer zone. Different Fired Heater configurations can be modeled or customized using tee, mixer, and heat exchanger unit operations.
- A pressure-flow specification option on each side and pass realistically models flow through Fired Heater operation according to the pressure gradient in the entire pressure network of the plant. Possible flow reversal situations can therefore be modeled.
- A comprehensive heat calculation inclusive of radiant, convective, and conduction heat transfer on radiant zone enables the prediction of process fluid temperature, Fired Heater wall temperature, and flue gas temperature.
- A dynamic model which accounts for energy and material holdups in each zone. Heat transfer in each zone depends on the flue gas properties, tube and Fired Heater wall properties, surface properties of metal, heat loss to the ambient, and the process stream physical properties.
- A combustion model which accounts for imperfect mixing of fuel, and allows automatic flame ignition or extinguished based on the oxygen availability in the fuel air mixture.

4.3.1 Theory

Combustion Reaction

The combustion reaction in the burner model of the Fired Heater performs pure hydrocarbon (C_xH_y) combustion calculations only. The extent of the combustion depends on the availability of oxygen which is usually governed by the air to fuel ratio.

Air to fuel ratio (*AF*) is defined as follows:

$$AF = \frac{\left(\frac{Mass\ of\ flow\ O_2}{\Sigma\ Mass\ flow\ of\ fuel}\right)}{Mass\ Ratio\ of\ O_2\ in\ Air} \tag{4.19}$$

You can set the combustion boundaries, such as the maximum *AF* and the minimum *AF*, to control the burner flame. The flame cannot light if the calculated air to fuel ratio falls below the specified minimum air to fuel ratio. The minimum air to fuel ratio and the maximum air to fuel ratio can be found on the Parameters page of the Design tab.

The heat released by the combustion process is the product of molar flowrate, and the heat of formation of the products minus the heat of formation of the reactants at combustion temperature and pressure. In the Fired Heater unit operation, a traditional reaction set for the combustion reactions is not required. You can choose the fuels components (the hydrocarbons and hydrogen) to be considered in the combustion reaction. You can see the mixing efficiency of each fuel component on the Parameter page of the Design tab.

Heat Transfer

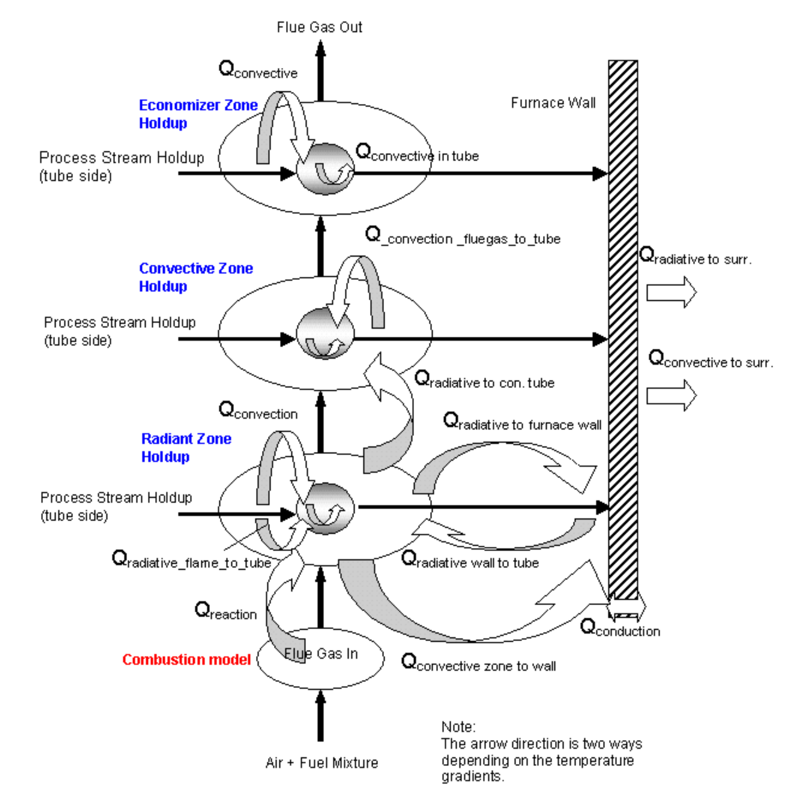
The Fired Heater heat transfer calculations are based on energy balances for each zone. The shell side of the Fired Heater contains five holdups:

- three in the radiant zone
- a convective zone
- an economizer zone holdup as outlined previously in **Figure 4.33**.

For the tube side, each individual stream passing through the respective zones is considered as a single holdup.

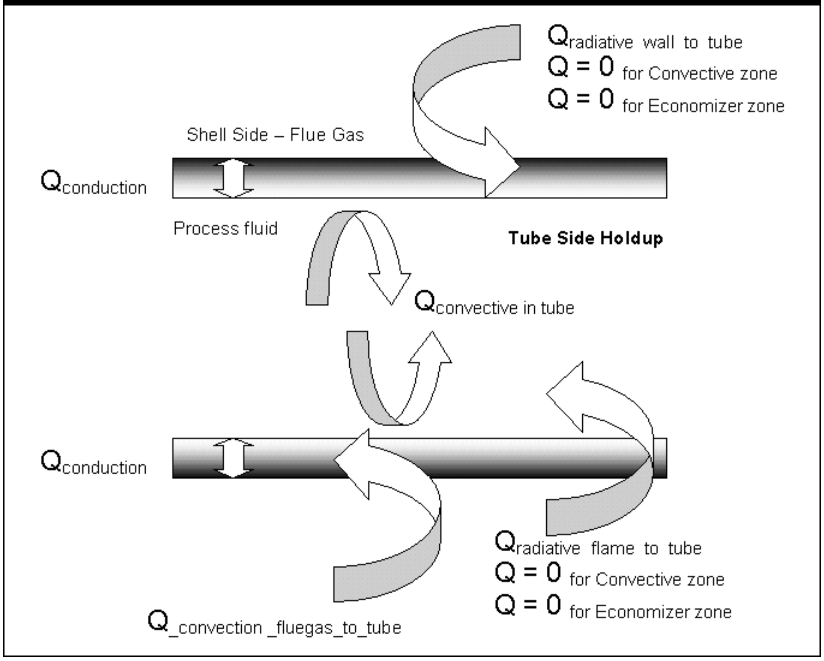
Major heat terms underlying the Fired Heater model are illustrated in the figure below.

Figure 4.34



The heat terms related to the tubeside are illustrated in the figure below.

Figure 4.35



Taking Radiant zone as an envelope, the following energy balance equation applies:

$$\frac{d(M_{rad}H_{rad})}{dt} + \frac{d(M_{RPFTube}H_{RPFTube})}{dt}$$
$$= (M_{RPF}H_{RPF})_{IN} - (M_{RPF}H_{RPF})_{OUT} + (M_{FG}H_{FG})_{IN}$$
$$- (M_{FG}H_{FG})_{OUT} - Q_{RadToCTube} - Q_{rad\ wall\ sur} - Q_{con\ wall\ sur} \tag{4.20}$$
$$+ Q_{rad\ wall\ to\ tube} - Q_{con\ to\ wall} + Q_{reaction}$$

where:

$\frac{d(M_{rad}H_{rad})}{dt}$ = *energy accumulation in radiant zone holdup shell side*

$\frac{d(M_{RPFTube}H_{RPFTube})}{dt}$ = *energy accumulation in radiant zone process fluid holdup (tube side)*

$(M_{RPF}H_{RPF})_{IN}$ = *total heat flow of process fluid entering radiant zone tube*

$(M_{RPF}H_{RPF})_{OUT}$ = *total heat flow of process fluid exiting radiant zone tube*

$(M_{FG}H_{FG})_{IN}$ = *total heat flow of fuel gas entering radiant zone*

$(M_{FG}H_{FG})_{OUT}$ = *total heat flow of fuel gas exiting radiant zone*

$Q_{RadToCTube}$ = *radiant heat of radiant zone to convective zone's tube bank*

$Q_{rad\_wall\_sur}$ = *radiant heat loss of Fired Heater wall in radiant zone to surrounding*

$Q_{con\_wall\_sur}$ = *convective heat loss of Fired Heater wall in radiant zone to surrounding*

$Q_{rad\_wall\_to\_tube}$ = *radiant heat from inner Fired Heater wall to radiant zone's tube bank*

$Q_{rad\_flame\_wall}$ = *radiant heat from flue gas flame to inner Fired Heater wall*

$Q_{con\_to\_wall}$ = *convective heat from flue gas to Fired Heater inner wall*

$Q_{reaction}$ = *heat of combustion of the flue gas*

Radiant Heat Transfer

For a hot object in a large room, the radiant energy emitted is given as:

$$Q_{radiative} = \delta A \varepsilon (T_1^{\ 4} - T_2^{\ 4}) \tag{4.21}$$

where:

δ = *Stefan-Boltzmann constant, 5.669x10*$^{-8}$ *W/m*^{2}*K*^{4}

ε = *emissivity, (0-1), dimensionless*

A = *area exposed to radiant heat transfer, m*^{2}

T_1 = *temperature of hot surface 1, K*

T_2 = *temperature of hot surface 2, K*

Convective Heat Transfer

The convective heat transfer taking part between a fluid and a metal is given in the following:

$$Q_{convective} = UA(T_1 - T_2) \tag{4.22}$$

where:

U = *overall heat transfer coefficient,* W/m^2K

A = *area exposed to convective heat transfer,* m^2

T_1 = *temperature of hot surface 1,K*

T_2 = *temperature of surface 2, K*

The *U* actually varies with flow according to the following *flow-U* relationship if this Flow Scaled method is used:

$$U_{used} = U_{specified}\left(\frac{Mass\ flow\ at\ time\ t}{Reference\ Mass\ flow}\right)^{0.8} \tag{4.23}$$

where:

$U_{specified}$ = *U value at steady state design conditions.*

The ratio of mass flow at time t to reference mass flow is also known as flow scaled factor. The minimum flow scaled factor is the lowest value, which the ratio is anticipated at low flow region. For the Fired Heater operation, the minimum flow scaled factor can be expressed only as a positive value.

For example, if the minimum flow scaled factor is +0.001 (0.1%), when this mass flow ratio is achieved, the U_{used} stays as a constant value. Therefore,

$$U_{used} = U_{specified}(0.001)^{0.8} \tag{4.24}$$

Conductive Heat Transfer

Conductive heat transfer in a solid surface is given as:

$$Q_{conductive} = -kA\frac{(T_1 - T_2)}{\Delta t} \tag{4.25}$$

where:

k = thermal conductivity of the solid material, W/mK

Δt *= thickness of the solid material, m*

A = area exposed to conductive heat transfer, m^2

T_1 *= temperature of inner solid surface 1, K*

T_2 *= temperature of outer solid surface 2, K*

Pressure Drop

The pressure drop across any pass in the Fired Heater unit operation can be determined in one of two ways:

- Specify the pressure drop - delta P.
- Define a pressure flow relation for each pass by specifying a k-value

If the pressure flow option is chosen for pressure drop determination in the Fired Heater pass, a *k* value is used to relate the frictional pressure drop and molar flow, *F* through the Fired Heater. This relation is similar to the general valve equation:

$$F = k\sqrt{\rho(P_1 - P_2)} \tag{4.26}$$

This general flow equation uses the pressure drop across the Fired Heater pass without any static head contribution. The quantity, (P_1-P_2) is defined as the frictional pressure loss which is used to “size” the flow.

The *k* value is calculated based on two criteria:

- If the flow of the system is larger than the value at k_{ref} (*k* reference flow), the *k* value remain unchanged. It is recommended that the *k* reference flow is taken as 40% of steady state design flow for better pressure flow stability at low flow range.
- If the flow of the system is smaller than the k_{ref}, the *k* value is given by:

$$k_{used} = k_{user\ specified} \times Factor \tag{4.27}$$

where:

Factor = value is determined by HYSYS internally to take into consideration the flow and pressure drop relationship for low flow regions.

The effect of k_{ref} is to increase the stability by modeling a more linear relationship between flow and pressure. This is also more realistic at low flows.

Dynamic Specifications

The following is a list of the minimum specifications required for the Fired Heater operation to solve:

Dynamic Specifications	Description
Connections	At least one radiant zone inlet stream and the respective outlet zone, one burner fuel/air feed stream and one combustion product stream must be defined. There is a minimum of one inlet stream and one outlet stream required per zone. Complete the connections group for each zone of the Design tab.
(Zone) Sizing	The dimensions of the tube and shell in each zone in the Fired Heater must be specified. All information in the Sizing page of the Rating tab must be completed.

Dynamic Specifications	Description
Heat Transfer	For each zone, almost all parameters in the Radiant Zone Properties group and Radiant/Convective/ Economizer Tube Properties groups are required except the Inner/Outer Scaled HX Coefficient.
Nozzle	Nozzle elevation is defaulted to 0. Elevation input is required when static head contribution option in Integrator property view is selected.
Pressure Drop	Either specify an overall delta P or an overall K value for the Fired Heater. Specify the pressure drop calculation method on the Tube Side PF page and Flue Gas PF page of the Dynamics tab.

4.3.2 Fired Heater Property View

There are two ways that you can add a Fired Heater to your simulation:

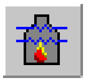
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Heat Transfer Equipment** radio button.
3. From the list of available unit operations, select Fired Heater.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

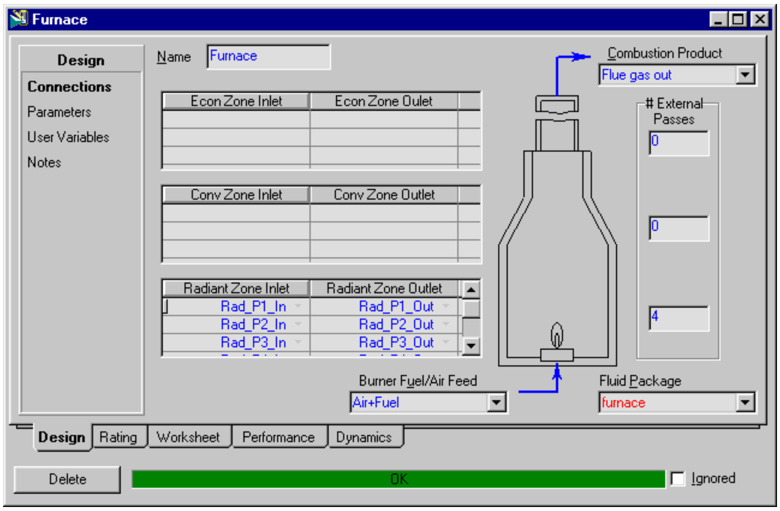
   You can also open the Object Palette by pressing **F4**.
2. Double-click the **Fired Heater** icon.



Fired Heater icon

The Fired Heater property view appears.

Figure 4.36



4.3.3 Design Tab

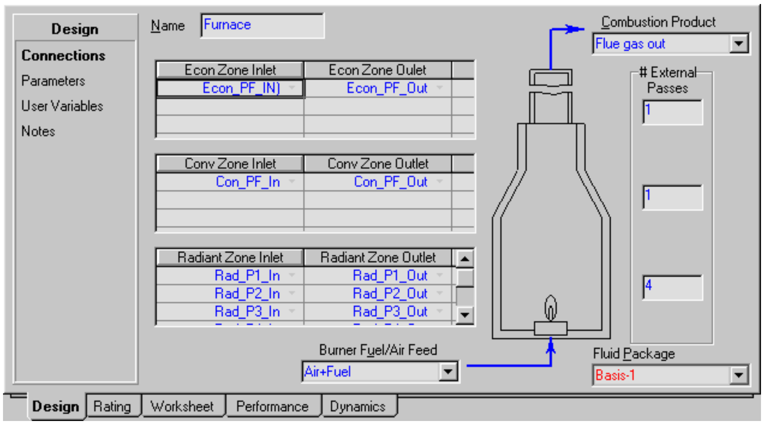
The Design tab contains the following pages:

- Connections
- Parameters
- User Variables
- Notes

Connections Page

On the Connections page, you can specify the name of the operation, and inlet and outlet streams.

Figure 4.37

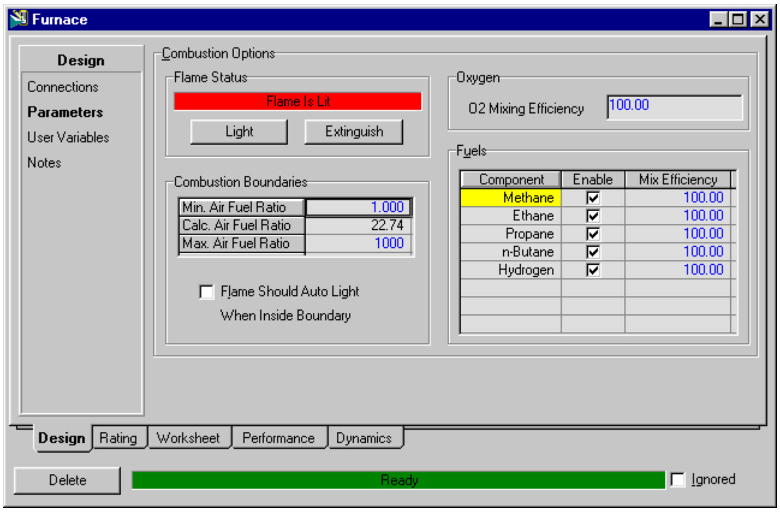


Object	Description
Econ Zone Inlet/ Outlet	You can specify multiple inlet and outlet streams for the Economizer zone.
Conv Zone Inlet/ Outlet	You can specify multiple inlet and outlet streams for the Convective zone.
Radiant Zone Inlet/ Outlet	You can specify multiple inlet and outlet streams for the Radiant zone.
Burner Fuel/Air Feed	Specifies the stream to be used for the burner fuel.
Combustion Product	The stream that contains the products from the combustion.
# External Passes	You can define the number of zones required by the Fired Heater

Parameters Page

The Parameters page is used to specify the Fired Heater combustion options.

Figure 4.38



This page is divided into four groups. The Flame Status group, along with displaying the flame status, allows you to toggle between a lit flame and an extinguished flame. The Oxygen group simply allows you to specify the oxygen mixing efficiency. The Combustion Boundaries group is used to set the combustion boundary based on a range of air fuel ratios. The checkbox, when active, allows you to auto-light the flame if your calculated air fuel ratio is within the boundary. Finally the Fuels group allows you to select the components present in your fuel as well as set their mixing efficiencies.

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor that allows you to record any comments or information regarding the specific unit operation, or the simulation case in general.

4.3.4 Rating Tab

The Rating tab contains the following pages:

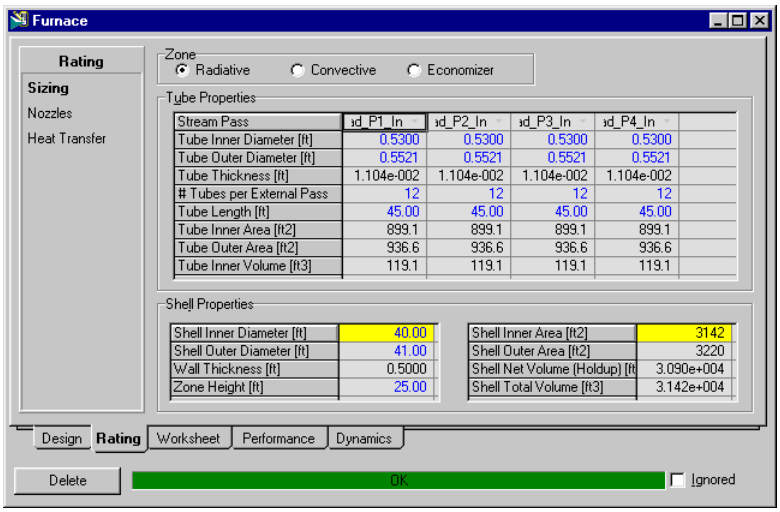
- Sizing
- Nozzles
- Heat Transfer

Each page is discussed in the following sections.

Sizing Page

On the Sizing page, you can specify the geometry of the radiant, convective, and economizer zones in the Fired Heater.

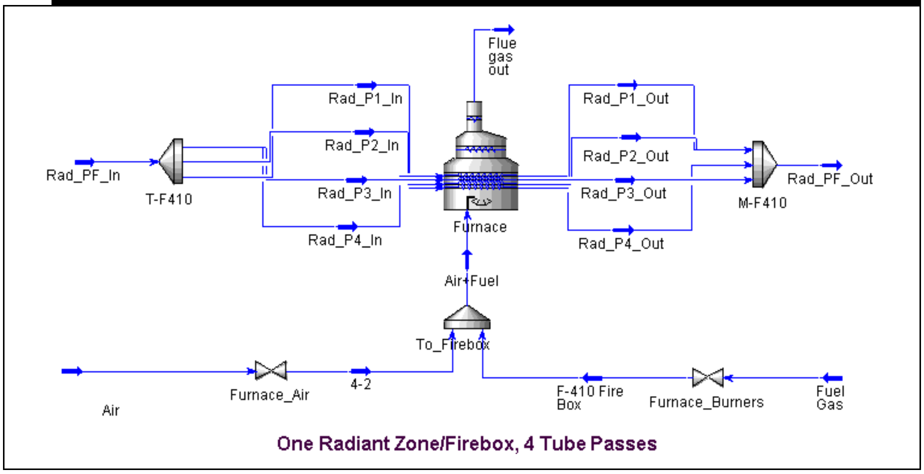
Figure 4.39



From the Zone group on the Sizing page, you can choose between Radiative, Convective, and Economizer zone property views by selecting the appropriate radio button. These property views contain information regarding the tube and shell properties. To edit or enter parameters within these property views, click the individual cell and make the necessary changes.

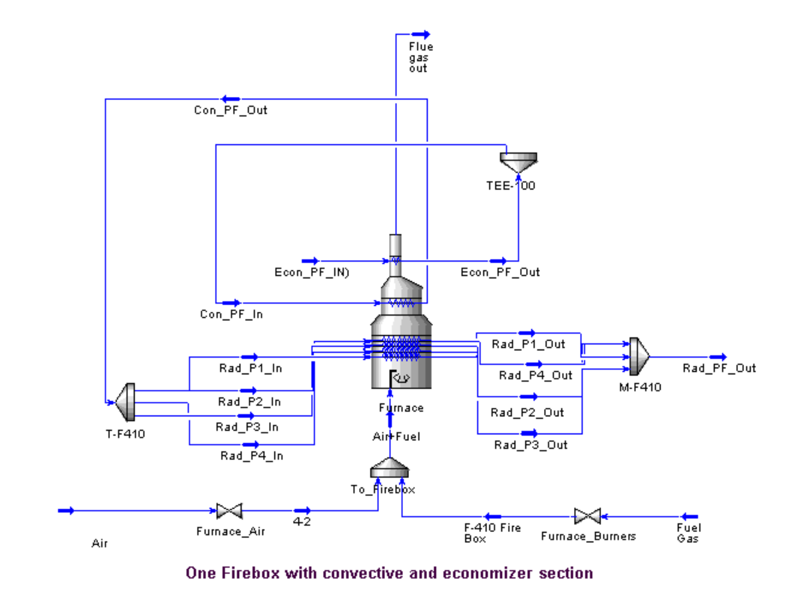
The figure below shows an example of the Fired Heater setup with one radiant zone/firebox only with four tube passes. This is the simplest type.

Figure 4.40



The figure below shows an example of the Fired Heater setup with a radiant, convective and economizer section.

Figure 4.41



Tube Properties Group

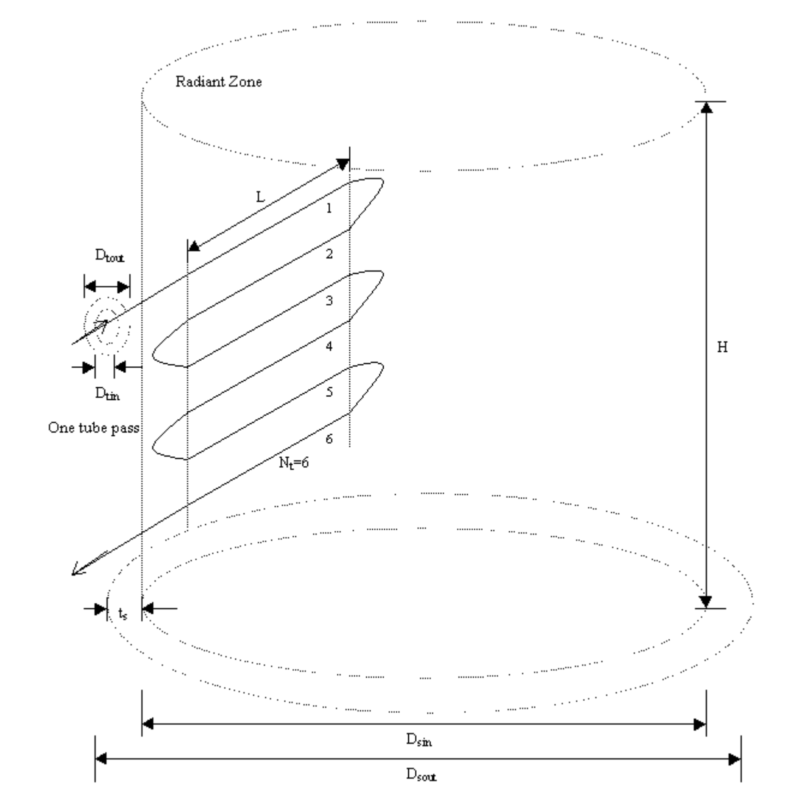
The Tube Properties group displays the following information regarding the dimension of the tube:

- stream pass
- tube inner diameter, D_{in}
- tube outer diameter, D_{out}
- tube thickness
- # tubes per external pass
- tube length, L
- tube inner area
- tube outer area
- tube inner volume

A pass in the Fired Heater is defined as a path where the process fluid flows through a distinctive inlet nozzle and outlet nozzle.

The figure below illustrates the various dimensions of the tube and shell.

Figure 4.42



Shell Properties Group

The Shell Properties group displays the following information regarding the dimension of the shell:

- shell inner diameter, D_{sin}
- shell outer diameter, D_{sout}
- wall thickness, t_s
- zone height, H
- shell inner area
- shell outer area
- shell net volume

- shell total volume

Nozzles Page

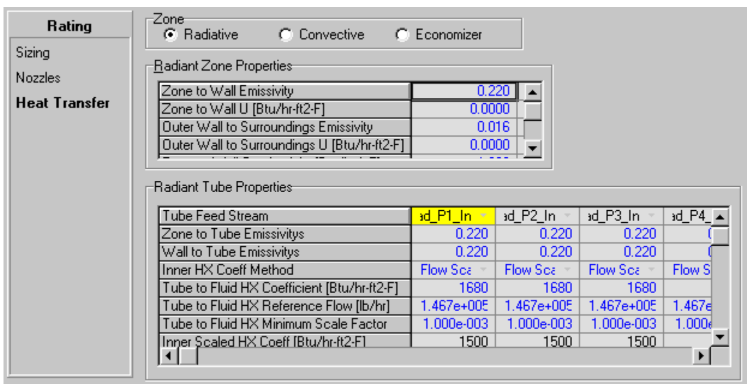
Figure 4.43

The information provided in the Nozzles page is applicable only in Dynamic mode. You can define the base elevation to ground level of the Fired Heater in the Nozzles page.

Heat Transfer Page

The information provided in the Heat Loss page is applicable only in Dynamic mode. This page displays the radiant heat transfer properties, heat transfer coefficients of the Fired Heater wall and tube, and shell area, tube area, and volume in each individual zone.

Figure 4.44



HYSYS accounts for the convective, conduction, and radiative heat transfer in the radiant zone. For the convective heat transfer calculation, you have two options:

- **User Specified**. You can specify the heat transfer coefficient of the inner tube and the outer tube.

- **Flow Scaled**. The heat transfer coefficient is scaled based on a specified flow.

The scaled heat transfer coefficient is defined by **Equation (4.23)**.

The same equation applies to the outer tube heat transfer coefficient calculation. Currently, the heat transfer coefficient *U* must be specified by the user. HYSYS calculates the heat transfer coefficient from the geometry/configuration of the Fired Heater. The radiant box or the fire box is assumed cylindrical in geometry.

Radiant Zone Properties Group

The following table describes each the parameters listed in the Radiant Zone group.

Radiant Zone Parameter	Description
Zone to Wall Emissivity	Emissivity of flue gas. HYSYS uses a constant value.
Zone to Wall U	Convective heat transfer coefficient of the radiative zone to the Fired Heater inner wall.
Outer Wall to Surrounding Emissivity	Emissivity of the Fired Heater outer wall.
Outer Wall to Surroundings U	Convective heat transfer coefficient of the Fired Heater outer wall to ambient.
Furnace Wall Conductivity / Specific Heat / Wall Density	These are user specified properties of a single layer of Fired Heater wall.

The Radiant, Convective, and Economizer Tube Properties groups all contain similar parameters, which are described in the following table.

Tube Properties	Description
Zone to Tube Emissivity	Emissivity of flue gas at radiant/convective zone to the tube in radiant/convective zone respectively.
Wall to Tube Emissivity	Radiant zone Fired Heater wall emissivity to the radiant zone tubes.

Tube Properties	Description
Inner HX Coeff Method	There are two options to calculate the Heat transfer coefficient in the tube: User Specified or Flow Scaled. Flow Scaled provides a more realistic HX calculation where: $$U_{used} = U_{specified}\left(\frac{mass}{mass_{ref}}\right)^{0.8}$$
Tube to Fluid HX Coefficient	Heat transfer coefficient of the tube to the process fluid.
Tube to Fluid HX Reference Flow	Mass flow at which the tube to fluid HX coefficient is based on. Usually the ideal steady state flow is recommended as input.
Tube to Fluid HX Minimum Scale Factor	The ratio of mass flow of the process fluid to the reference mass flow in the tube. The valve ranges from a value of zero to one. If the process flow in the tube becomes less than the scale factor, the heat transfer coefficient used is smaller than U specified.
Inner Scaled HX Coefficient	The HX coefficient obtained if the Flow Scaled (U_{used}) method is applied to perform the calculation.
Tube C_p, Density, Conductivity	Metal properties of the tube in their respective zones.
Outer HX Coefficient Method	Method used to calculate the shell side HX coefficient. Two options available: User Specified or Flow Scaled.
Zone to Tube HX Coefficient	HX coefficient in the radiative/convective/ economizer or flue gas zones to the respective tubes.
Zone to Tube HX Reference Flow	Mass flow of the flue gas at which the outer HX coefficient is based upon. This is usually designed using the ideal steady state flow of the flue gas.
Zone to Tube HX Minimum Scale Factor	Mass ratio of flue gas flow to the flue gas reference mass flow. This value ranges from zero to one. If the process flow in the tubes is less than this value, the HX coefficient used is set to zero.
Outer Scaled U	The actual HX coefficient used in the calculation if the Flow Scaled option is selected.

In general the Tube to Fluid HX Coefficient is always shown in a common Fired Heater flowsheet, however, the Zone to Wall U and Outer Wall to Surroundings U are usually unknown. The Outer wall to Surroundings U can be easily estimated from the Fired Heater convective heat loss calculation, **Equation (4.22)** if the total heat loss via Fired Heater wall is known. The total heat loss is normally expressed as a percentage of total Fired Heater duty. A 3-5% heat loss is an acceptable estimate.

Estimating Zone to Wall U requires trial and error techniques. Enter a value of U then observe the temperature profile of the flue gas exiting the radiant zone.

4.3.5 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the heat exchanger unit operation.

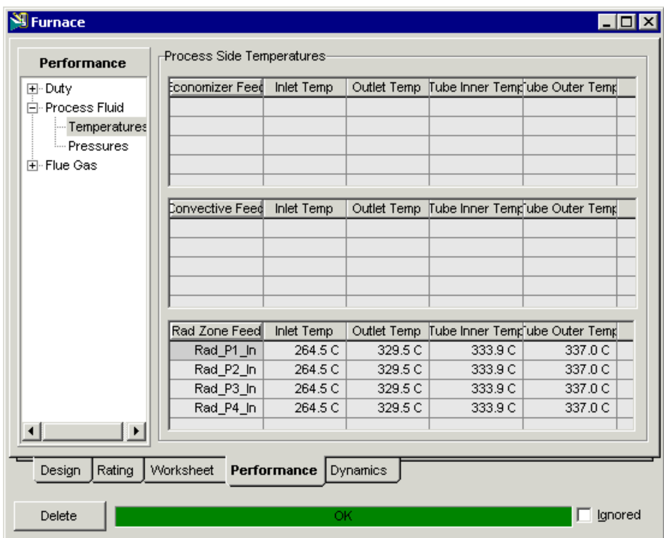
To view the stream parameters broken down per stream phase, open the Worksheet tab of the stream property view.

The PF Specs page is relevant to dynamics cases only.

4.3.6 Performance Tab

The performance tab contains three pages which highlight the calculated temperature, duty, and pressure of the Fired Heater operation.

Figure 4.45

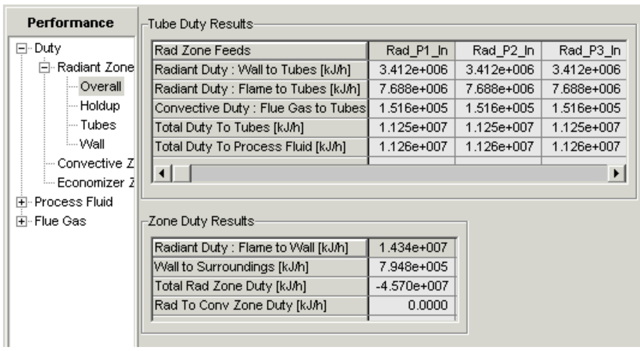


Duty Page

The Duty page displays the results of the Fired Heater energy balance calculation. The Duty page contains three levels/ branches: Radiant Zone, Convective Zone, and Economizer Zone.

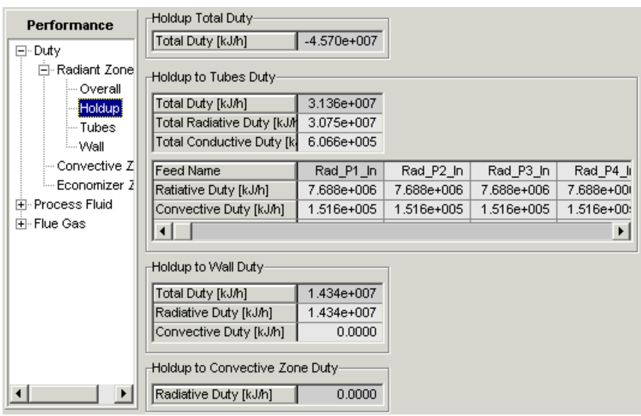
- If you select **Radiant Zone** from the tree browser, the following four levels/branches containing information regarding the Tube Duty results and Zone Duty results appear:
  - Overall

Figure 4.46



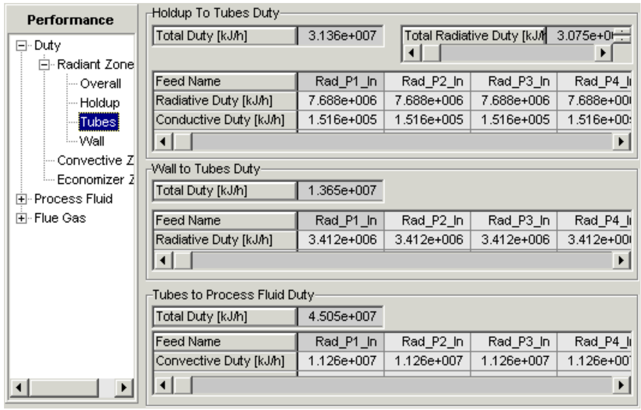
  - Holdup

Figure 4.47



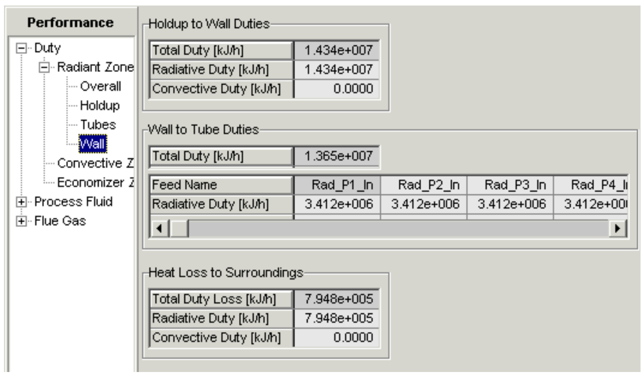
- Tubes

Figure 4.48



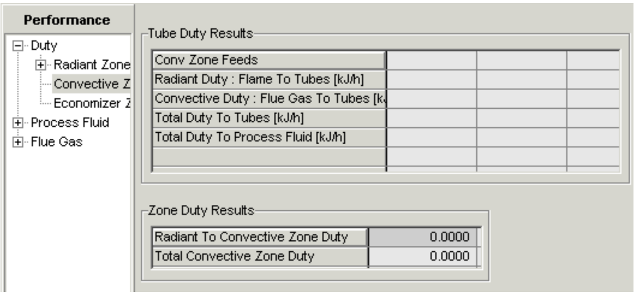
- Wall

Figure 4.49



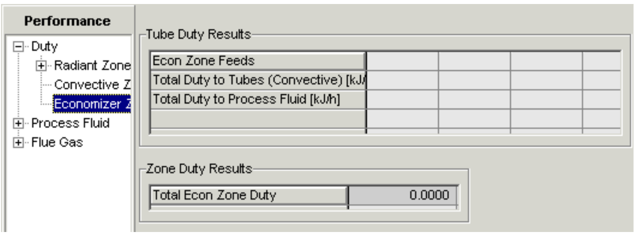
- If you select the **Convective Zone** from the tree browser, the following parameters from the Tube Duty Results group and the Zone Duty Results group appear:

Figure 4.50

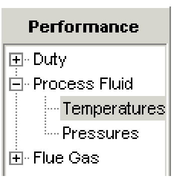


- If you select the **Economizer Zone** from the tree browser, the following parameters from the Tube Duty results group and Zone Duty results group appear:

Figure 4.51



Process Fluid Page



Sub pages on the Process Fluid page.

The Process Fluid page contains two sub-pages:

- Temperatures
- Pressures

In the Temperatures sub-page, the following parameters appear:

- Inlet Temp, Inlet stream process fluid temperature
- Outlet Temp, Outlet stream process fluid temperature
- Tube Inner Temp, Tube inner wall temperature

In the Pressures sub-page, the following parameters appear:

- Inlet pressure, inlet stream pressure
- Friction Delta P, friction pressure drop across the tube
- Static Head Delta P, static pressure of the stream
- Outlet Pressure, outlet stream pressure

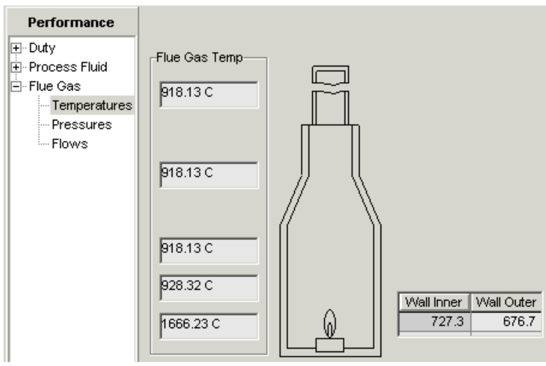
Flue Gas Page

The Flue Gas page contains the following sub-pages:

- Temperatures
- Pressures
- Flows

On the Temperatures sub-page, you can view your flue gas temperature and Fired Heater inner/outer wall temperatures.

Figure 4.52



Similarly, the Pressures sub-page displays the flue gas pressures, frictional delta P, and static head delta P. The Flow sub-page displays the flue gas molar/mass flow.

4.3.7 Dynamics Tab

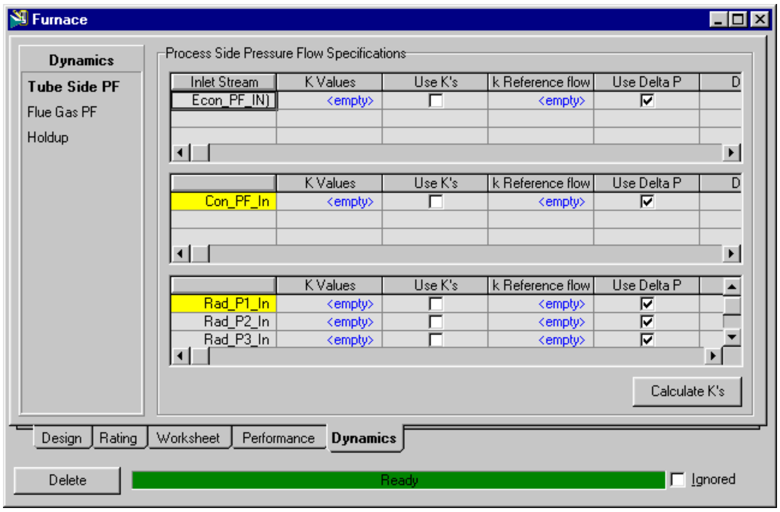
The Dynamics tab contains information pertaining to pressure specifications for he dynamic calculations. The information is sorted into the following pages:

- Tube Side PF
- Flue Gas PF
- Holdup

Tube Side PF Page

The Tube Side PF page allows you to specify how the pressure drop in each pass is calculated.

Figure 4.53



The following table outlines the tube side PF options available on this page.

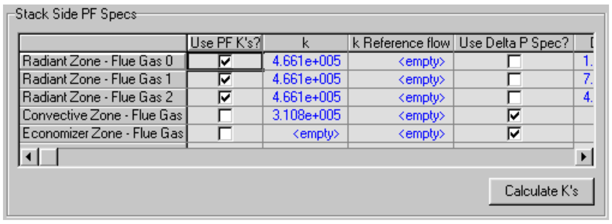
Option	Description
Use K's?	If this checkbox is selected, the K method is used to calculate Delta P across the pass.

Option	Description
Use Delta P Spec?	If this checkbox is selected, the pressure drop is fixed at this specified value.
Calculate K's	If this button is clicked, HYSYS calculates the K required to maintain a specified Delta P across a defined flow condition.

Flue Gas PF Page

On the Flue Gas PF page, you can specify how the pressure drop in each pass is calculated.

Figure 4.54



The following table outlines the tube side PF options available on this page.

Option	Description
Use PF K's	If this checkbox is selected, the K method is used to calculate Delta P across the pass.
Use Delta P	If this checkbox is selected, the pressure drop is fixed at this specified value.
Calculate K's	If this button is clicked, HYSYS calculates the K required to maintain a specified Delta P across a defined flow condition.

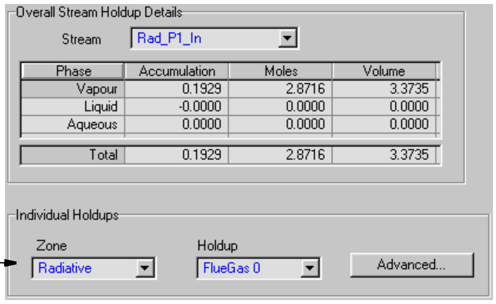
Holdup Page

Refer to **Section 1.3.3 - Holdup Page** for more information.

The Holdup page contains information regarding each stream's holdup properties and composition.

Figure 4.55

The Individual Holdups group contains two drop-down lists (**Zone** and **Holdup**) that enable you to select and view information on individual zone and holdup section.



4.4 Heat Exchanger

The Heat Exchanger performs two-sided energy and material balance calculations. The Heat Exchanger is very flexible, and can solve for temperatures, pressures, heat flows (including heat loss and heat leak), material stream flows, or UA.

Additional Heat Exchanger models, such as TASC and STX, are also available. Contact your local AspenTech representative for details.

In HYSYS, you can choose the Heat Exchanger Model for your analysis. Your choices include an End Point analysis design model, an ideal (Ft=1) counter-current Weighted design model, a steady state rating method, and a dynamic rating method for use in dynamic simulations. The dynamic rating method is available as either a Basic or Detailed model, and can also be used in Steady State mode for Heat Exchanger rating. The unit operation also allows the use of third party Heat Exchanger design methods via OLE Extensibility.

The following are some of the key features of the dynamic Heat Exchanger operation:

- A pressure-flow specification option which realistically models flow through the Heat Exchanger according to the pressure network of the plant. Possible flow reversal situations can therefore be modeled.

In Dynamic mode, the shell and tube of the Heat Exchanger is capable of storing inventory like other dynamic vessel operations. The direction of flow of material through the Heat Exchanger is governed by the pressures of the surrounding unit operations.

- The choice between a Basic and Detailed Heat Exchanger model. Detailed Heat Exchanger rating information can be used to calculate the overall heat transfer coefficient and pressure drop across the Heat Exchanger.
- A dynamic holdup model which calculates level in the Heat Exchanger shell based on its geometry and orientation.
- A heat loss model which accounts for the convective and conductive heat transfer that occurs across the Heat Exchanger shell wall.

4.4.1 Theory

The Heat Exchanger calculations are based on energy balances for the hot and cold fluids.

Steady State

In the following general relations, the hot fluid supplies the Heat Exchanger duty to the cold fluid:

$$\text{Balance Error} = (M_{cold}[H_{out} - H_{in}]_{cold} - Q_{leak}) - (M_{hot}[H_{in} - H_{out}]_{hot} - Q_{loss}) \tag{4.28}$$

where:

M = fluid mass flow rate

H = enthalpy

Q_{leak} = *heat leak*

Q_{loss} = *heat loss*

Balance Error = a Heat Exchanger Specification that equals zero for most applications

$_{hot}$ and $_{cold}$ = *hot and cold fluids*

$_{in}$ and $_{out}$ = *inlet and outlet stream*

The Heat Exchanger operation allows the heat curve for either side of the exchanger to be broken into intervals. Rather than calculating the energy transfer based on the terminal conditions of the exchanger, it is calculated for each of the intervals, then summed to determine the overall transfer.

The total heat transferred between the tube and shell sides (Heat Exchanger duty) can be defined in terms of the overall heat transfer coefficient, the area available for heat exchange, and the log mean temperature difference:

$$Q = UA\Delta T_{LM}F_t \tag{4.29}$$

where:

U = *overall heat transfer coefficient*

A = *surface area available for heat transfer*

ΔT_{TM} = *log mean temperature difference (LMTD)*

F_t = *LMTD correction factor*

The heat transfer coefficient and the surface area are often combined for convenience into a single variable referred to as UA. The LMTD and its correction factor are defined in the Performance section.

Dynamic

The following general relation applies to the shell side of the Basic model Heat Exchanger.

$$M_{shell}(H_{in} - H_{out})_{shell} - Q_{loss} + Q = \rho\frac{d(VH_{out})_{shell}}{dt} \tag{4.30}$$

For the tube side:

$$M_{tube}(H_{in} - H_{out})_{tube} - Q = \rho\frac{d(VH_{out})_{tube}}{dt} \tag{4.31}$$

where:

M_{shell} = *shell fluid flow rate*

M_{tube} = *tube fluid flow rate*

ρ = *density*

H = *enthalpy*

Q_{loss} = *heat loss*

Q = *heat transfer from the tube side to the shell side*

V = *volume shell or tube holdup*

Refer to **Section 1.3.4 - Heat Loss Model** in the **HYSYS Dynamic Modeling** guide for more information.

The term Q_{loss} represents the heat lost from the shell side of the dynamic Heat Exchanger. For more information regarding how Q_{loss} is calculated.

Pressure Drop

The pressure drop of the Heat Exchanger can be determined in one of three ways:

- Specify the pressure drop.
- Calculate the pressure drop based on the Heat Exchanger geometry and configuration.
- Define a pressure flow relation in the Heat Exchanger by specifying a k-value.

If the pressure flow option is chosen for pressure drop determination in the Heat Exchanger, a k value is used to relate the frictional pressure loss and flow through the exchanger. This relation is similar to the general valve equation:

$$f = \sqrt{density} \times k\sqrt{P_1 - P_2} \tag{4.32}$$

This general flow equation uses the pressure drop across the Heat Exchanger without any static head contributions. The quantity, P_1 - P_2, is defined as the frictional pressure loss which is used to “size” the Heat Exchanger with a k-value.

Dynamic Specifications

The following tables list the minimum specifications required for the Heat Exchanger unit operation to solve in Dynamic mode.

The Basic Heat Exchanger model requires the following dynamic specifications:

Specification	Description
Volume	The tube and shell volumes must be specified.
Overall UA	The Overall UA must be specified.
Pressure Drop	Either specify an Overall Delta P or an Overall K-value for the Heat Exchanger. Specify the Pressure Drop calculation method in the Dynamic Specifications group on the Specs page of the Dynamics tab. You can also specify the Overall Delta P values for the shell and tube sides on the Sizing page of the Rating tab.

The Detailed Heat Exchanger model requires the following dynamic specifications:

Specification	Description
Sizing Data	The tube and shell sides of the Heat Exchanger must be completely specified on the Sizing page of the Rating tab. The overall tube/shell volumes, and the heat transfer surface area are calculated from the shell and tube ratings information.
Overall UA	Either specify an Overall UA or have it calculated from the Shell and Tube geometry. Specify the U calculation method on the Parameters page of the Rating tab. The U calculation method can also be specified on the Model page of the Dynamics tab.
Pressure Drop	Either specify an Overall Delta P or an Overall K-value for the Heat Exchanger. Specify the Pressure Drop calculation method on the Parameters page of the Rating tab. You can also specify the Pressure Drop calculation method in the Pressure Flow Specifications group on the Specs page of the Dynamics tab.

4.4.2 Heat Exchanger Property View

There are two ways that you can add a Heat Exchanger to your simulation:

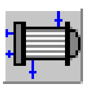
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Heat Transfer Equipment** radio button.
3. From the list of available unit operations, select Heat Exchanger.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

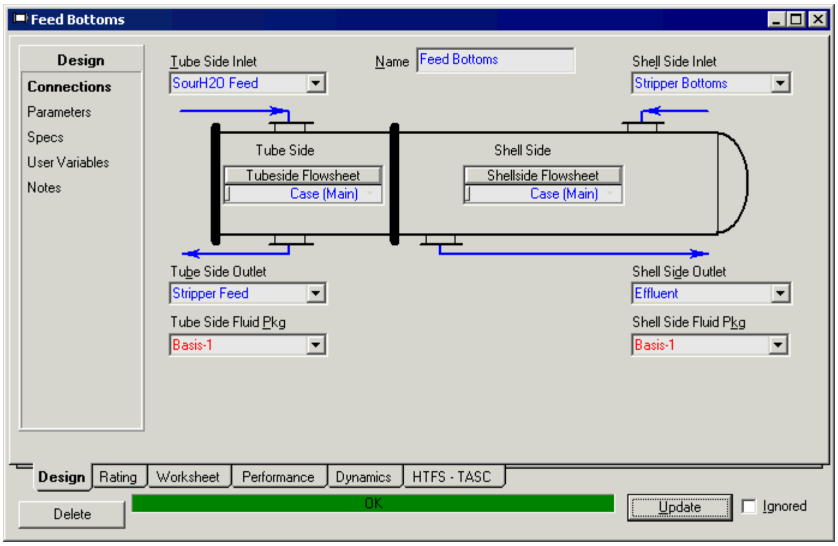
   You can also open the Object Palette by pressing **F4**.
2. Double-click the **Heat Exchanger** icon.



Heat Exchanger icon

The Heat Exchanger property view is displayed.

Figure 4.56



The **Update** button enables you to update the heat exchanger calculation when in Dynamic mode. For example, if you make a configurational change to the heat exchanger, click this button to reset the equations around the heat exchanger before running the simulation calculation in Dynamic mode.

4.4.3 Design Tab

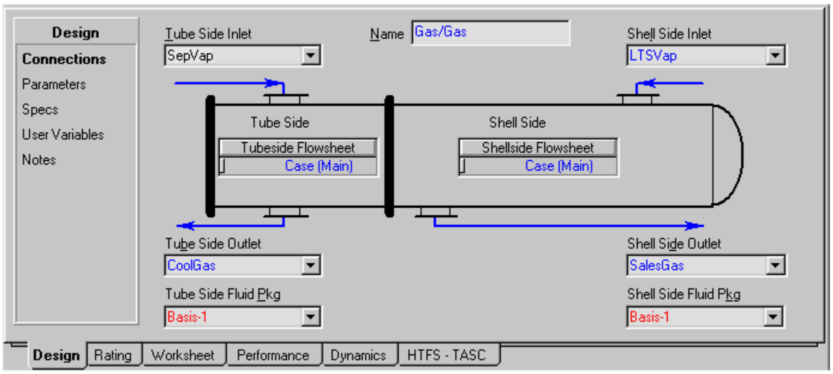
The Design tab contains the following pages:

- Connections
- Parameters
- Specs
- User Variables
- Notes

Connections Page

The Connections page allows you to specify the operation name, and the inlet and outlet streams of the shell and tube.

Figure 4.57



The main flowsheet is the default flowsheet for the Tube and Shell side. You can select a subflowsheet on the Tube and/or Shell side which allows you to choose inlet and outlet streams from that flowsheet. This is useful for processes such as the Refrigeration cycle, which require separate fluid packages for each side. You can define a subflowsheet with a different fluid package, and then connect to the main flowsheet Heat Exchanger.

Parameters Page

The Parameters page allows you to select the Heat Exchanger Model and specify relevant physical data. The parameters appearing on the Parameters page depend on which Heat Exchanger Model you select.

When a heat exchanger is installed as part of a column subflowsheet (available when using the Modified HYSIM Inside-Out solving method) these Heat Exchanger Models are not available. Instead, in the column subflowsheet, the heat exchanger is “Calculated from Column” as a simple heat and mass balance.

From the Heat Exchanger Model drop-down list, select the calculation model for the Heat Exchanger. The following Heat Exchanger models are available:

- Exchanger Design (Endpoint)
- Exchanger Design (Weighted)
- Steady State Rating
- Dynamic Rating
- HTFS - Engine
- TASC Heat Exchanger

Refer to the **TASC Thermal Reference** guide for more information.

The HTFS - Engine and TASC Heat Exchanger options are only available if you have installed TASC.

For both the Endpoint and Weighted models, you can specify whether your Heat Exchanger experiences heat leak/loss.

- **Heat Leak**. Loss of cold side duty due to leakage. Duty gained to reflect the increase in temperature.
- **Heat Loss**. Loss of hot side duty due to leakage. Duty lost to reflect the decrease in temperature.

The table below describes the radio buttons in the Heat Leak/ Loss group of the Endpoint and Weighted models.

Radio Button	Description
None	By default, the None radio button is selected.
Extremes	On the hot side, the heat is considered to be “lost” where the temperature is highest. Essentially, the top of the heat curve is being removed to allow for the heat loss/leak. This is the worst possible scenario. On the cold side, the heat is gained where the temperature is lowest.
Proportional	The heat loss is distributed over all of the intervals.

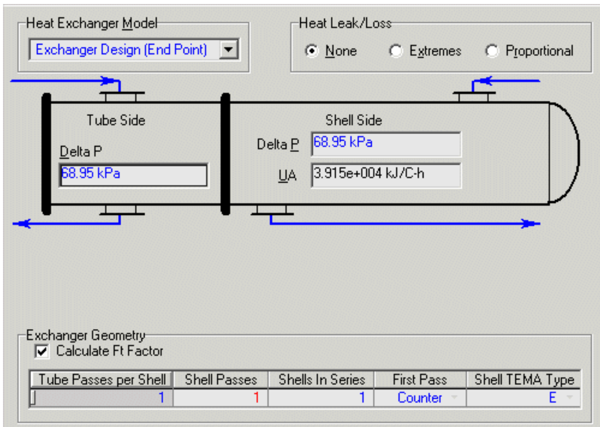
Refer to **Section 4.4.4 - Rating Tab** for further details.

All Heat Exchanger models allow for the specification of either Counter or Co-Current tube flow.

End Point Model

The End Point model is based on the standard Heat Exchanger duty equation (**Equation (4.29)**) defined in terms of overall heat transfer coefficient, area available for heat exchange, and the log mean temperature difference (LMTD).

Figure 4.58



The main assumptions of the model are as follows:

- Overall heat transfer coefficient, U is constant.
- Specific heats of both shell and tube side streams are constant.

The End Point model treats the heat curves for both Heat Exchanger sides as linear. For simple problems where there is no phase change and C_p is relatively constant, this option may be sufficient to model your Heat Exchanger. For non-linear heat flow problems, the Weighted model should be used instead.

The following parameters are available when the End Point model is selected:

Parameters	Description
Tubeside and Shellside Delta P	The pressure drops (DP) for the tube and shell sides of the exchanger can be specified here. If you do not specify the Delta P values, HYSYS calculates them from the attached stream pressures.
UA	The product of the Overall Heat Transfer Coefficient, and the Total Area available for heat transfer. The Heat Exchanger duty is proportional to the log mean temperature difference, where UA is the proportionality factor. The UA can either be specified, or calculated by HYSYS.
Exchanger Geometry	The Exchanger Geometry is used to calculate the Ft Factor using the End Point Model. It is not available for the weighted model. Refer to the Rating tab for more information on the Exchanger Geometry.

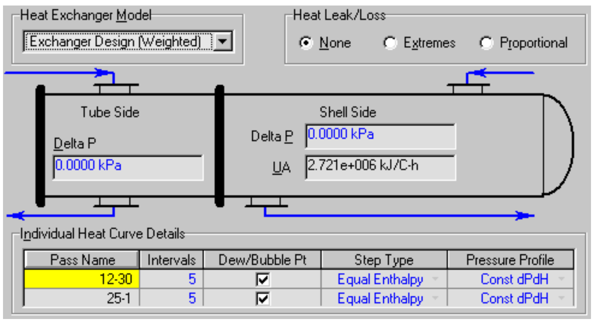
Weighted Model

The Weighted model is an excellent model to apply to non-linear heat curve problems such as the phase change of pure components in one or both Heat Exchanger sides. With the Weighted model, the heating curves are broken into intervals, and an energy balance is performed along each interval. A LMTD and UA are calculated for each interval in the heat curve, and summed to calculate the overall exchanger UA.

The Weighted model is available only for counter-current exchangers, and is essentially an energy and material balance model. The geometry configurations which affect the Ft correction factor are not taken into consideration in this model.

When you select the Weighted model, the Parameters page appears as shown in the figure below.

Figure 4.59



The following table describes the parameters available on the Parameters page when the Weighted model is selected:

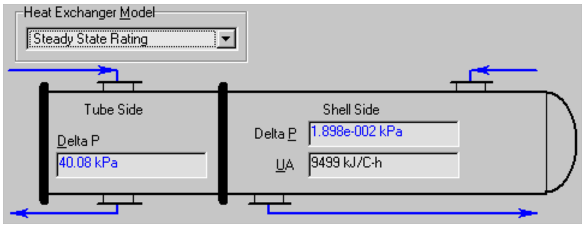
Parameters	Description
Tubeside and Shellside Delta P	The pressure drops (DP) for the tube and shell sides of the exchanger can be specified here. If you do not specify the DP values, HYSYS calculates them from the attached stream pressures.
UA	The product of the Overall Heat Transfer Coefficient and the Total Area available for heat transfer. The Heat Exchanger duty is proportional to the log mean temperature difference, where UA is the proportionality factor. The UA can either be specified, or calculated by HYSYS.
Individual Heat Curve Details	For each side of the Heat Exchanger, the following parameters appear (all but the Pass Names can be modified). • **Pass Name**. Identifies the shell and tube side according to the names you provided on the Connections page. • **Intervals**. The number of intervals can be specified. For non-linear temperature profiles, more intervals are necessary. • **Dew/Bubble Point**. Select this checkbox to add a point to the heat curve for the dew and/or bubble point. If there is a phase change occurring in either pass, the appropriate checkbox should be selected. There are three choices for the **Step Type**: • **Equal Enthalpy**. All intervals have an equal enthalpy change. • **Equal Temperature**. All intervals have an equal temperature change. • **Auto Interval**. HYSYS determines where points should be added to the heat curve. This is designed to minimize error using the least number of intervals. The **Pressure Profile** is updated in the outer iteration loop, using one of the following methods: • **Constant dPdH**.Maintains constant dPdH during update. • **Constant dPdUA**.Maintains constant dPdUA during update. • **Constant dPdA**. Maintains constant dPdA during update. This is not currently applicable to the Heat Exchanger, as the area is not predicted. • **Inlet Pressure**.Pressure is constant and equal to the inlet pressure. • **Outlet Pressure**. Pressure is constant and equal to the outlet pressure.

Steady State Rating Model

The Steady State Rating model is an extension of the End Point model to incorporate a rating calculation, and uses the same assumptions as the End Point model. If you provide detailed geometry information, you can rate the exchanger using this model. As the name suggests, this model is only available for steady state rating.

When dealing with linear or nearly linear heat curve problems, the Steady State Rating model should be used. Due to the solver method incorporated into this rating model, the Steady State Rating model can perform calculations exceptionally faster than the Dynamic Rating model.

Figure 4.60



The following parameters are available on the Parameters page when the Steady State Rating model is selected:

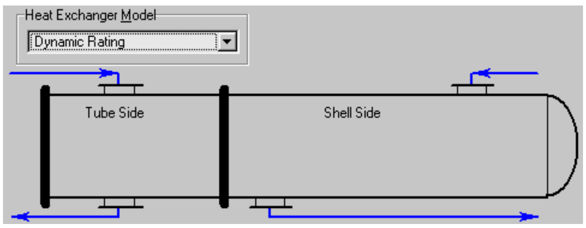
Parameters	Description
Tubeside and Shellside Delta P	The pressure drops (DP) for the tube and shell sides of the exchanger can be specified here. If you do not specify the Delta P values, HYSYS calculates them from the attached stream pressures.
UA	The product of the Overall Heat Transfer Coefficient, and the Total Area available for heat transfer. The Heat Exchanger duty is proportional to the log mean temperature difference, where UA is the proportionality factor. The UA can either be specified, or calculated by HYSYS.

Dynamic Rating

Two models are available for Dynamic Rating using the Heat Exchanger unit operation: a Basic and a Detailed model. If you specify three temperatures or two temperatures and a UA, you can rate the exchanger with the Basic model. If you provide detailed geometry information, you can rate the exchanger using the Detailed model.

The Specs page no longer appears when Dynamic Rating is selected.

Figure 4.61



The Basic model is based on the same assumptions as the End Point model, which uses the standard Heat Exchanger duty equation (**Equation (4.29)**) defined in terms of overall heat transfer coefficient, area available for heat exchange, and the log mean temperature difference. The Basic model is actually the counterpart of the End Point model for dynamics and dynamic rating. The Basic model can also be used for steady state Heat Exchanger rating.

The Detailed model is based on the same assumptions as the Weighted model, and divides the Heat Exchanger into a number of heat zones, performing an energy balance along each interval. This model requires detailed geometry information about your Heat Exchanger. The Detailed model is actually the counterpart of the Weighted model for dynamics and dynamic rating, but can also be used for steady state Heat Exchanger rating.

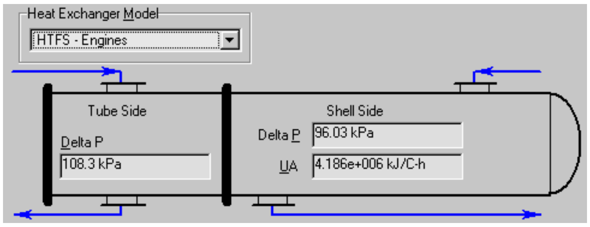
The Basic and Detailed Dynamic Rating models share rating information with the Dynamics Heat Exchanger model. Any rating information entered using these models is observed in Dynamic mode.

Once the Dynamic Rating model is selected, no further information is required on the Parameters page of the Design tab. You can choose the model (Basic or Detailed) on the Parameters page of the Rating tab.

HTFS - Engine

The figure below shows the Parameters page of the Design tab, if you select the HTFS - Engine model. Notice that the values in the fields appear in black, indicating that they are HYSYS calculated values, and you cannot change them in the current fields.

Figure 4.62



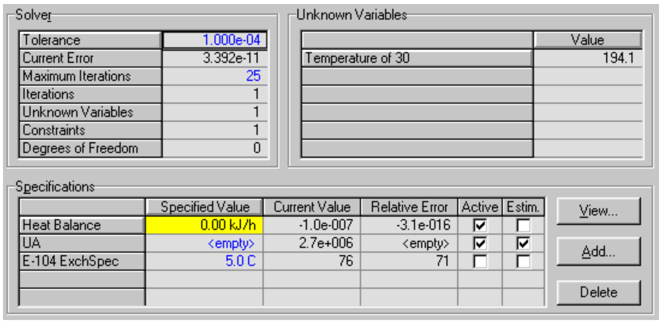
To change the variable values shown on this page, you have to go to the HTFS - TASC tab on the Heat Exchanger property view. Refer to **Section 4.4.8 - HTFS-TASC Tab** for more information.

Specs Page

The Specs page includes three groups that organize various specifications and solver information. The information provided on the Specs page is only valid for the Weighted, Endpoint, and Steady State Rating models.

If you are working with a Dynamic Rating model, the Specs page does not appear on the Design tab.

Figure 4.63



Solver Group

The following parameters are listed in the Solver group:

Parameters	Details
Tolerance	The calculation error tolerance can be set.
Current Error	When the current error is less than the calculation tolerance, the solution is considered to have converged.
Iterations	The current iteration of the outer loop appears. In the outer loop, the heat curve is updated and the property package calculations are performed. Non-rigorous property calculations are performed in the inner loop. Any constraints are also considered in the inner loop.

Unknown Variables Group

HYSYS lists all unknown Heat Exchanger variables according to your specifications. Once the unit has solved, the values of these variables appear.

Specifications Group

The Heat Balance (specified at 0 kJ/h) is considered to be a constraint.

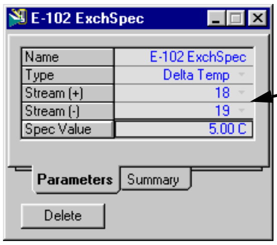
Without the Heat Balance specification, the heat equation is not balanced.

This is a Duty Error spec, which you cannot turn off. Without the Heat Balance specification, you could, for example, completely specify all four Heat Exchanger streams, and have HYSYS calculate the Heat Balance error which would be displayed in the Current Value column of the Specifications group.

The UA is also included as a default specification. HYSYS displays this as a convenience, since it is a common specification. You can either use this spec or deactivate it.

You can view or delete highlighted specifications by using the buttons at the right of the group. A specification property view appears automatically each time a new spec is created via the Add button. The figure below shows a typical property view of a specification, which is accessed via the **View** or **Add** button.

Figure 4.64



Defining the Delta Temp spec requires two stream names and a value for the specification.

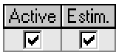
Each specification property view has the following tabs:

- Parameters
- Summary

The Summary page is used to define whether the specification is Active or an Estimate. The Spec Value is also shown on this page.

Information specified on the specification property view also appears in the Specifications group.

All specifications are one of the following three types:

Specification Type	Description
Active 	An active specification is one that the convergence algorithm is trying to meet. An active specification always serves as an initial estimate (when the Active checkbox is selected, HYSYS automatically selects the Estimate checkbox). An active specification exhausts one degree of freedom. An Active specification is one that the convergence algorithm is trying to meet. An Active specification is on when both checkboxes are selected.
Estimate 	An Estimate is considered an Inactive specification because the convergence algorithm is not trying to satisfy it. To use a specification as an estimate only, clear the **Active** checkbox. The value then serves only as an initial estimate for the convergence algorithm. An estimate does not use an available degree of freedom. An Estimate is used as an initial “guess” for the convergence algorithm, and is considered to be an inactive specification.
Completely Inactive 	To disregard the value of a specification entirely during convergence, clear both the **Active** and **Estimate** checkboxes. By ignoring rather than deleting a specification, it remains available if you want to use it later. A Completely Inactive specification is one that is ignored completely by the convergence algorithm, but can be made Active or an Estimate at a later time.

The specification list allows you to try different combinations of the above three specification types. For example, suppose you have a number of specifications, and you want to determine which ones should be active, which should be estimates and which ones should be ignored altogether. By manipulating the checkboxes among various specifications, you can test various combinations of the three types to see their effect on the results.

The available specification types include the following:

Specification	Description
Temperature	The temperature of any stream attached to the Heat Exchanger. The hot or cold inlet equilibrium temperature can also be defined. • The Hot Inlet Equilibrium temperature is the temperature of the inlet hot stream minus the heat loss temperature drop. • The Cold Inlet Equilibrium temperature is the temperature of the inlet cold stream plus the heat leak temperature rise.
Delta Temp	The temperature difference at the inlet or outlet between any two streams attached to the Heat Exchanger. The hot or cold inlet equilibrium temperatures (which incorporate the heat loss/heat leak with the inlet conditions) can also be used.
Minimum Approach	Minimum internal temperature approach. The minimum temperature difference between the hot and cold stream (not necessarily at the inlet or outlet).
UA	The overall UA (product of overall heat transfer coefficient and heat transfer area).
LMTD	The overall log mean temperature difference.
Duty	The overall duty, duty error, heat leak or heat loss. The duty error should normally be specified as 0 so that the heat balance is satisfied. The heat leak and heat loss are available as specifications only if the Heat Loss/Leak is set to Extremes or Proportional on the Parameters page.
Duty Ratio	A duty ratio can be specified between any two of the following duties: overall, error, heat loss, and heat leak.
Flow	The flowrate of any attached stream (molar, mass or liquid volume).
Flow Ratio	The ratio of the two inlet stream flowrates. All other ratios are either impossible or redundant (in other words, the inlet and outlet flowrates on the shell or tube side are equal).

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor that allows you to record any comments or information regarding the specific unit operation or the simulation case in general.

4.4.4 Rating Tab

The Rating tab contains the following pages:

- Sizing
- Parameters

The Parameters page is used exclusively by the dynamics Heat Exchanger, and only becomes active either in Dynamic mode or while using the Dynamic Rating model.

- Nozzles
- Heat Loss

Sizing Page

The Sizing page provides Heat Exchanger sizing related information. Based on the geometry information, HYSYS is able to calculate the pressure drop and the convective heat transfer coefficients for both Heat Exchanger sides and rate the exchanger.

The information is grouped under three radio buttons:

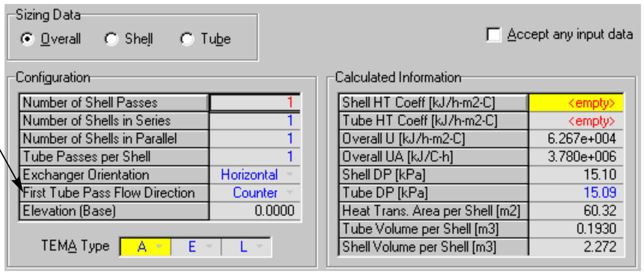
- Overall
- Shell
- Tube

Overall

When you select the Overall radio button, the overall Heat Exchanger geometry appears:

Figure 4.65

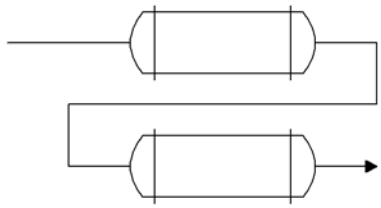
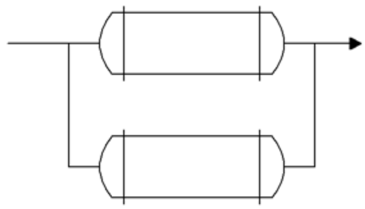
Tube flow direction can be defined as either Counter or Co-Current for all heat exchanger calculation models.



In the Configuration group, you can specify whether multiple shells are used in the Heat Exchanger design.

The following fields appear, and can be modified in, the Configuration group.

Field	Description
Number of Shell Passes	You have the option of HYSYS performing the calculations for Counter Current (ideal with Ft = 1.0) operation, or for a specified number of shell passes. Specify the number of shell passes to be any integer between 1 and 7. When the shell pass number is specified, HYSYS calculates the LMTD correction factor (Ft) for the current exchanger design. A value lower than 0.8 generally corresponds to inefficient design in terms of the use of heat transfer surface. More passes or larger temperature differences should be used in this case. For **n** shell passes, HYSYS solves the heat exchanger on the basis that at least 2**n** tube passes exist. Charts for Shell and Tube Exchanger LMTD Correction Factors, as found in the GPSA Engineering Data Book, are normally in terms of **n** shell passes and 2**n** or more tube passes.

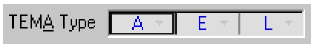
Field	Description
Number of Shells in Series	If a multiple number of shells are specified in series, the configuration is shown as follows: 
Number of Shells in Parallel	If a multiple number of shells are specified in parallel, the configuration is shown as follows:  Currently, multiple shells in parallel are not supported in HYSYS.
Tube Passes per Shell	The number of tube passes per shell. The default setting is 2 (in other words, the number of tubes equal to 2n, where n is the number of shells.)
Exchanger Orientation	The exchanger orientation defines whether or not the shell is horizontal or vertical. Used only in dynamic simulations. When the shell orientation is vertical, you can also specify whether the shell feed is at the top or bottom via the Shell Feed at Bottom checkbox. The Shell Feed at Bottom checkbox is only visible for the vertical oriented exchanger.
First Tube Pass Flow Direction	Specifies whether or not the tube feed is co-current or counter-current.
Elevation (base)	The height of the base of the exchanger above the ground. Used only in dynamic simulations.

You can specify the number of shell and tube passes in the shell of the Heat Exchanger. In general, at least 2n tube passes must be specified for every n shell pass. The exception is a counter-current flow Heat Exchanger which has 1 shell pass and one tube pass. The orientation can be specified as a vertical or horizontal Heat Exchanger. The orientation of the Heat Exchanger does not impact the steady state solver, however, it is used in the Dynamics Heat Exchanger Model in the calculation of liquid level in the shell.

For a more detailed discussion of TEMA-style shell-and-tube heat exchangers, refer to page 11-33 of the Perry's Chemical Engineers' Handbook (1997 edition).

The shape of Heat Exchanger can be specified using the TEMA-style drop-down lists. The first list contains a list of front end stationary head types of the Heat Exchanger. The second list contains a list of shell types. The third list contains a list of rear end head types.

Figure 4.66



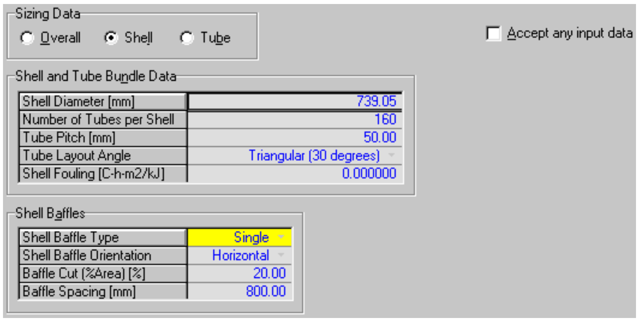
In the Calculated Information group, the following Heat Exchanger parameters are listed:

- Shell HT Coeff
- Tube HT Coeff
- Overall U
- Overall UA
- Shell DP
- Tube DP
- Heat Trans. Area per Shell
- Tube Volume per Shell
- Shell Volume per Shell

Shell

Selecting the Shell radio button allows you to specify the shell configuration and the baffle arrangement in each shell.

Figure 4.67



In the Shell and Tube Bundle Data group, you can specify whether multiple shells are used in the Heat Exchanger design. The following fields appear, and can be modified in, the Shell and Tube Bundle Data group.

Field	Description
Shell Diameter	Diameter of the shell(s).
Number of Tubes per Shell	Number of tubes per shell. You can change the value in this field.
Tube Pitch	Shortest distance between the centres of two adjacent tubes.
Tube Layout Angle	In HYSYS, the tubes in a single shell can be arranged in four different symmetrical patterns: • Triangular (30°) • Triangular Rotated (60°) • Square (90°) • Square Rotated (45°) For more information regarding the benefits of different tube layout angles, refer to page 139 of Process Heat Transfer by Donald Q. Kern (1965)
Shell Fouling	The shell fouling factor is taken into account in the calculation of the overall heat transfer coefficient, UA.

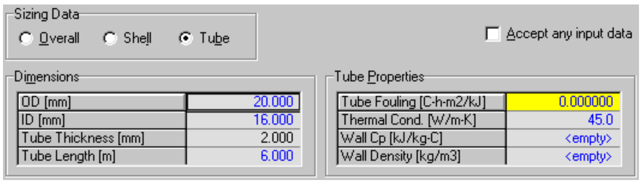
The following fields appear, and can be modified in, the Shell Baffles group:

Field	Description
Shell Baffle Type	You can choose from four different baffle types: • Single • Double • Triple • Grid
Shell Baffle Orientation	You can choose whether the baffles are aligned horizontally or vertically along the inner shell wall.
Baffle cut (Area%)	You can specify the percent area where the liquid flows through relative to the cross sectional area of the shell. The baffle cut is expressed as a percent of net free area. The net free area is defined as the total cross-sectional area in the flow direction parallel to the tubes minus the area blocked off by the tubes (essentially the percentage of open area).
Baffle Spacing	You can specify the space between each baffle.

Tube

Selecting the Tube radio button allows you to specify the tube geometry information in each shell.

Figure 4.68



The Dimensions group allows you to specify the following tube geometric parameters:

Field	Description
Outer Tube Diameter (OD) **Inner Tube Diameter (ID)** **Tube Thickness**	Two of the three listed parameters must be specified to characterize the tube width dimensions.
Tube Length	Heat transfer length of one tube in a single Heat Exchanger shell. This value is not the actual tube length.

In the Tube Properties group, the following metal tube heat transfer properties must be specified:

- Tube Fouling Factor
- Thermal Conductivity
- Wall Specific Heat Capacity, Cp
- Wall Density

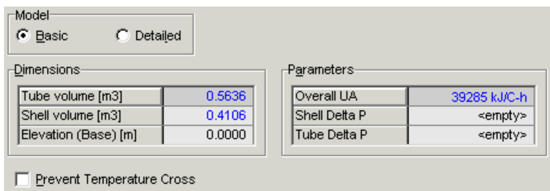
Parameters Page

The Parameters page of the Rating tab is used to define rating parameters for the Dynamic Rating model. On the Parameters page, you can specify either a Basic model or a Detailed model. For the Basic model, you must define the Heat Exchanger overall UA and pressure drop across the shell and tube. For the Detailed model, you must define the geometry and heat transfer parameters of both the shell and tube sides in the Heat Exchanger operation. In order for either the Basic or Detailed Heat Exchanger Model to completely solve, the Parameters page must be completed.

Basic Model

When you select the Basic model radio button on the Parameters page in Dynamic mode, the following property view appears.

Figure 4.69



The Dimensions group contains the following information:

- Tube Volume
- Shell Volume
- Elevation (Base)

The tube volume, shell volume, and heat transfer area are calculated from Shell and Tube properties specified by selecting the Shell and Tube radio buttons on the Sizing page. The elevation of the base of the Heat Exchanger can be specified but does not impact the steady state solver.

The **Prevent Temperature Cross** checkbox is used to activate additional model options when selected. These additional model options prevent temperature crosses by automatically reducing the heat transfer rate slowly.

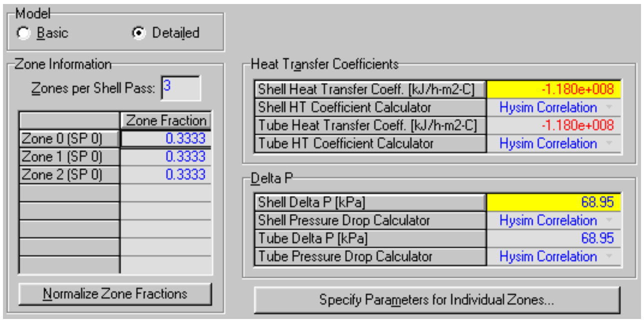
The Parameters group includes the following Heat Exchanger parameters. All but the correction factor, F, can be modified:

Field	Description
Overall UA	The product of the Overall Heat Transfer Coefficient, and the Total Area available for heat transfer. The Heat Exchanger duty is proportional to the log mean temperature difference, where UA is the proportionality factor. The UA can either be specified, or calculated by HYSYS.
Tubeside and Shellside Delta P	The pressure drops (DP) for the tube and shell sides of the exchanger can be specified here. If you do not specify the DP values, HYSYS calculates them from the attached stream pressures.

Detailed Model

The Detailed model option allows you to specify the zone information, heat transfer coefficient, and Delta P details. When you select the Detailed model radio button on the Parameters page, the following property view appears.

Figure 4.70



Zone Information

HYSYS can partition the Heat Exchanger into discrete multiple sections called zones. Because shell and tube stream conditions do not remain constant across the operation, the heat transfer parameters are not the same along the length of the Heat Exchanger. By dividing the Heat Exchanger into zones, you can make different heat transfer specifications for individual zones, and therefore more accurately model an actual Heat Exchanger.

In the Zone Information group you can specify the following:

Field	Description
Zones per Shell Pass	Enter the number of zones you want for one shell. The total number of zones in a Heat Exchanger shell is calculated as: $$Total\ Zones = Total\ Shell\ Passes \cdot Zones$$
Zone Fraction	The fraction of space the zone occupies relative to the total shell volume. HYSYS automatically sets each zone to have the same volume. You can modify the zone fractions to occupy a larger or smaller proportion of the total volume. Click the Normalize Zone Fractions button in order to adjust the sum of fractions to equal one.

Heat Transfer Coefficients

The Heat Transfer Coefficients group contains information regarding the calculation of the overall heat transfer coefficient, UA, and local heat transfer coefficients for the fluid in the tube, h_i, and the fluid surrounding the tube, h_o. The heat transfer coefficients can be determined in one of two ways:

- The heat transfer coefficients can be specified using the rating information provided on the Parameters page and the stream conditions.
- You can specify the heat transfer coefficients.

For fluids without phase change, the local heat transfer coefficient, h_i, is calculated according to the Sieder-Tate correlation:

$$h_i = \frac{0.027k_m}{D_i}\left(\frac{D_iG_i}{\mu_i}\right)^{0.8}\left(\frac{C_{p,\ i}\mu_i}{k_m}\right)^{1/3}\left(\frac{\mu_i}{\mu_{i,\ w}}\right)^{0.14} \qquad (4.33)$$

where:

G_i = *mass velocity of the fluid in the tubes (velocity*density)*

μ_i = *viscosity of the fluid in the tube*

$\mu_{i,\ w}$ = *viscosity of the fluid inside tubes, at the tube wall*

$C_{p,i}$ = *specific heat capacity of the fluid inside the tube*

The relationship between the local heat transfer coefficients, and the overall heat transfer coefficient is shown in **Equation (4.34)**.

$$U = \frac{1}{\left[\frac{1}{h_o} + r_o + r_w + \frac{D_o}{D_i}\left(r_i + \frac{1}{h_i}\right)\right]} \qquad (4.34)$$

where:

U = *overall heat transfer coefficient*

h_o = *local heat transfer coefficient outside tube*

h_i = *local heat transfer coefficient inside tube*

r_o = fouling factor outside tube

r_i = fouling factor inside tube

r_w = tube wall resistance

D_o = outside diameter of tube

D_i = inside diameter of tube

The Heat Transfer coefficients group contains the following information:

Field	Description
Shell/Tube Heat Transfer Coefficient	The local Heat Transfer Coefficients, h_o and h_i, can be specified or calculated.
Shell/Tube HT Coefficient Calculator	The Heat Transfer Coefficient Calculator allows you to either specify or calculate the local Heat Transfer Coefficients. Specify the cell with one of following options: • **Shell & Tube**. The local heat transfer coefficients, h_o and h_i, are calculated using the heat exchange rating information and correlations. • **U specified**. The local heat transfer coefficients, h_o and h_i, are specified by you.

Delta P

The Delta P group contains information regarding the calculation of the shell and tube pressure drop across the exchanger. In Steady State mode, the pressure drop across either the shell or tube side of the Heat Exchanger can be calculated in one of two ways:

- The pressure drop can be calculated from the rating information provided in the Sizing page and the stream conditions.
- The pressure drop can be specified.

The Delta P group contains the following information:

Field	Description
Shell/Tube Delta P	The pressure drop across the Shell/Tube side of the Heat Exchanger can be specified or calculated.
Shell/Tube Delta P Calculator	The Shell/Tube Delta P Calculator allows you to either specify or calculate the shell/tube pressure drop across the Heat Exchanger. Specify the cell with one of following options: • **Shell & Tube Delta P Calculator**. The pressure drop is calculated using the Heat Exchanger rating information and correlations. • **User specified**. The pressure drop is specified by you. • **Non specified**. This option is only applicable in Dynamic mode. Pressure drop across the Heat Exchanger is calculated from a pressure flow relation.

Detailed Heat Model Properties

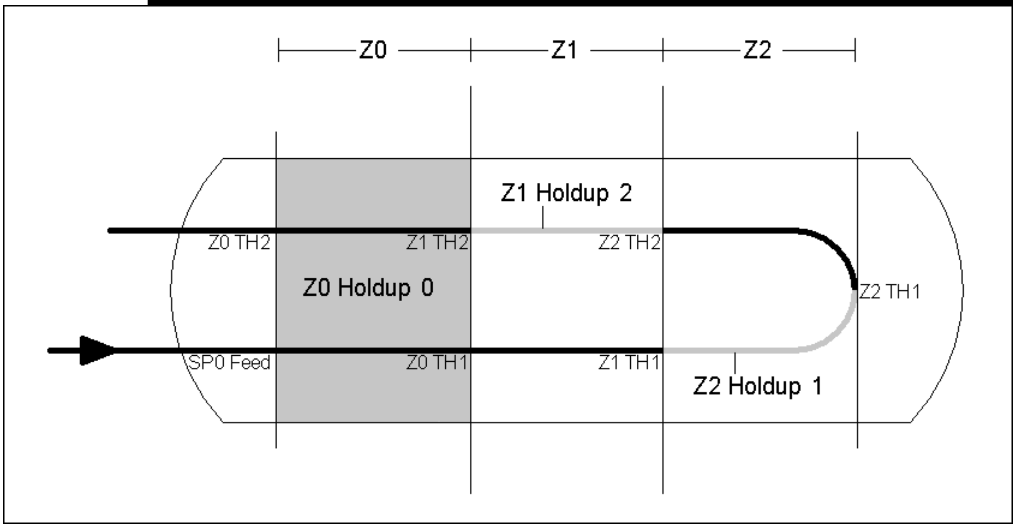
When you click the Specify Parameters for Individual Zones button, the Detailed Heat Model Properties property view appears. The Detailed Heat Model Properties property view displays the detailed heat transfer parameters and holdup conditions for each zone.

HYSYS uses the following terms to describe different locations within the Heat Exchanger.

Location Term	Description
Zone	HYSYS represents the zone using the letter “Z”. Zones are numbered starting from 0. For instance, if there are 3 zones in a Heat Exchanger, the zones are labeled: Z0, Z1, and Z2.
Holdup	HYSYS represents the holdup within each zone with the letter “H”. Holdups are numbered starting from 0. “Holdup 0” is always the holdup of the shell within the zone. Holdups 1 through n represents the n tube holdups existing in the zone.
Tube Location	HYSYS represents tube locations using the letters “TH”. Tube locations occur at the interface of each zone. Depending on the number of tube passes per shell pass, there can be several tube locations within a particular zone. For instance, 2 tube locations exist for each zone in a Heat Exchanger with 1 shell pass and 2 tube passes. Tube locations are numbered starting from 1.

Consider a shell and tube Heat Exchanger with 3 zones, 1 shell pass, and 2 tube passes. The following diagram labels zones, tube locations, and hold-ups within the Heat Exchanger:

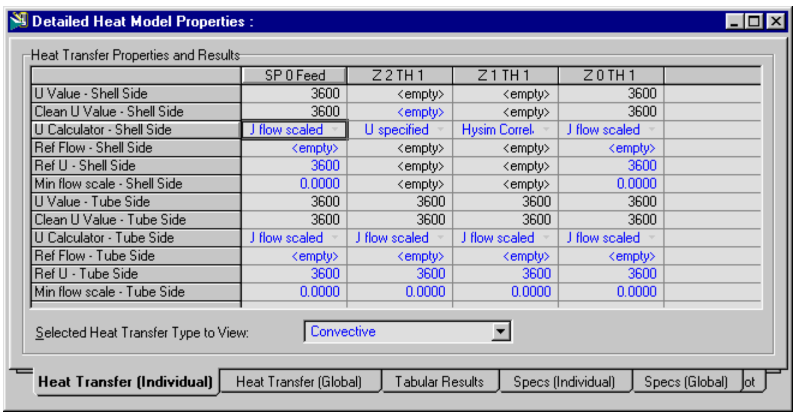
Figure 4.71



Heat Transfer (Individual) Tab

Information regarding the heat transfer elements of each tube location in the Heat Exchanger appears on the Heat Transfer (Individual) tab.

Figure 4.72



	SP 0 Feed	Z 2 TH 1	Z 1 TH 1	Z 0 TH 1
U Value - Shell Side	3600	<empty>	<empty>	3600
Clean U Value - Shell Side	3600	<empty>	<empty>	3600
U Calculator - Shell Side	J flow scaled	U specified	Hysim Correl.	J flow scaled
Ref Flow - Shell Side	<empty>	<empty>	<empty>	<empty>
Ref U - Shell Side	3600	<empty>	<empty>	3600
Min flow scale - Shell Side	0.0000	<empty>	<empty>	0.0000
U Value - Tube Side	3600	3600	3600	3600
Clean U Value - Tube Side	3600	3600	3600	3600
U Calculator - Tube Side	J flow scaled	J flow scaled	J flow scaled	J flow scaled
Ref Flow - Tube Side	<empty>	<empty>	<empty>	<empty>
Ref U - Tube Side	3600	3600	3600	3600
Min flow scale - Tube Side	0.0000	0.0000	0.0000	0.0000

Heat transfer from the fluid in the tube to the fluid in the shell occurs through a series of heat transfer resistances or elements. There are two convective elements, and one conductive element associated with each tube location.

This tab organizes all the heat transfer elements for each tube location in one spreadsheet. You can choose whether Conductive or Convective elements will appear by selecting the appropriate element type in the Heat Transfer Type drop-down list.

The following is a list of possible elements for each tube location:

Heat Transfer Element	Description
Convective Element	The Shell Side element is associated with the local heat transfer coefficient, h_o, around the tube. The Tube Side is associated with the local heat transfer coefficient, h_i, inside the tube.These local heat transfer coefficients can be calculated by HYSYS or modified by you.
Conductive Element	This element is associated with the conduction of heat through the metal wall of the tube. The conductivity of the tube metal, and the inside and outside metal wall temperatures appear. You can modify the conductivity.

Heat Transfer (Global) Tab

The Heat Transfer (Global) tab displays the heat transfer elements for the entire Heat Exchanger. You can choose whether the overall Conductive or Convective elements are to appear by selecting the appropriate element type in the Heat Transfer Type drop-down list.

Tabular Results Tab

The Tabular Results tab displays the following stream properties for the shell and tube fluid flow paths. The feed and exit stream conditions appear for each zone.

- Temperature
- Pressure
- Vapour Fraction

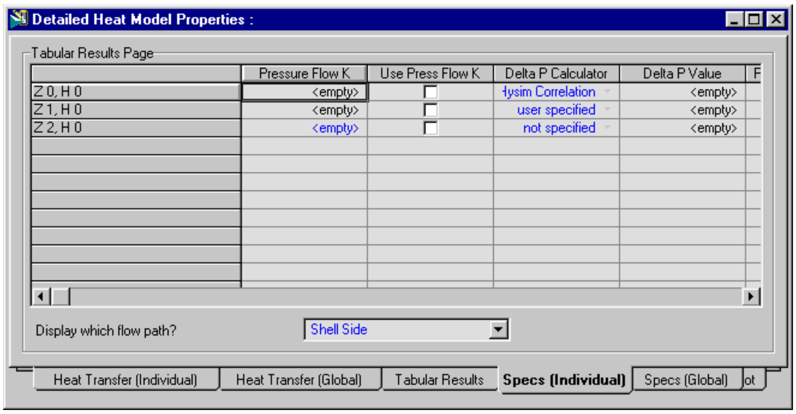
- Molar Flow
- Enthalpy
- Cumulative UA
- Cumulative Heat Flow
- Length (into Heat Exchanger)

You can choose whether the flow path is shell or tube side by selecting the appropriate flow path in the Display which flow path? drop-down list.

Specs (Individual) Tab

The Specs (Individual) tab displays the pressure drop specifications for each shell and tube holdup in one spreadsheet.

Figure 4.73



You can choose whether the shell or tube side appears by selecting the appropriate flow path in the Display which flow path? drop-down list.

The Pressure Flow K and Use Pressure Flow K columns are applicable only in Dynamic mode.

Specs (Global) Tab

The Specs (Global) tab displays the pressure drop specifications for the entire shell and tube holdups. The Pressure Flow K and Use Pressure Flow K columns are applicable only in Dynamic mode.

You can choose whether the shell or tube side appears by selecting the appropriate flow path in the Display which flow path? drop-down list.

Plots Tab

The information displayed on the Plots tab is a graphical representation of the parameters provided on the Tabular Results tab. You can plot the following variables for the shell and tube side of the Heat Exchanger:

- Vapour Fraction
- Molar Flow
- Enthalpy
- Cumulative UA
- Heat Flow
- Length

Nozzles Page

Refer to **Section 1.3.6 - Nozzles Page** for more information.

The Nozzles page contains information regarding the elevation and diameter of the nozzles.

The placement of feed and product nozzles on the Detailed Dynamic Heat Exchanger operation has physical meaning. The exit stream's composition depends on the exit stream nozzle's location and diameter in relation to the physical holdup level in the vessel. If the product nozzle is located below the liquid level in the vessel, the exit stream draws material from the liquid holdup. If the product nozzle is located above the liquid level, the exit stream draws material from the vapour holdup.

If the liquid level sits across a nozzle, the mole fraction of liquid in the product stream varies linearly with how far up the nozzle the liquid is.

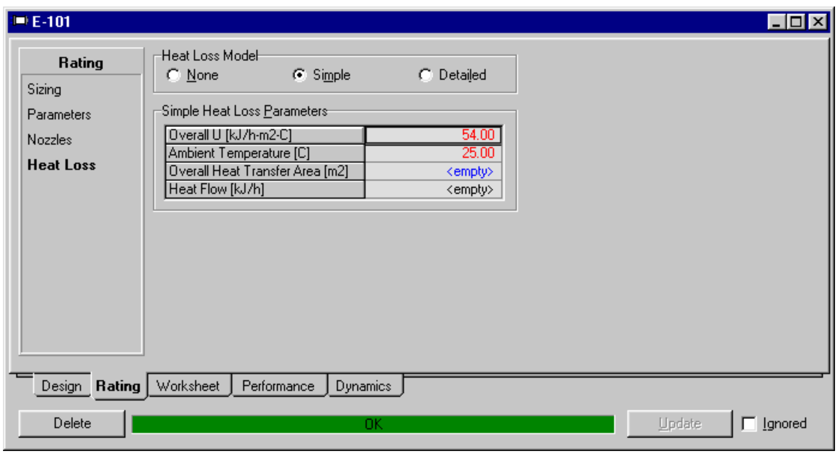
Essentially, all vessel operations in HYSYS are treated the same. The compositions and phase fractions of each product stream depend solely on the relative levels of each phase in the holdup and the placement of the product nozzles, so a vapour product nozzle does not necessarily produce pure vapour. A 3-phase separator may not produce two distinct liquid phase products from its product nozzles.

Heat Loss Page

The Heat Loss page contains heat loss parameters which characterize the amount of heat lost across the vessel wall. You can choose either to have no heat loss model, a Simple heat loss model or a Detailed heat loss model.

Simple Heat Loss Model

Figure 4.74



When you select the Simple radio button, the following parameters appear:

- Overall U
- Ambient Temperature
- Overall Heat Transfer Area
- Heat Flow

Detailed Heat Loss Model

Refer to **Section 1.6.1 - Detailed Heat Model** in the **HYSYS Dynamic Modeling** guide for more information.

The Detailed model allows you to specify more detailed heat transfer parameters. The HYSYS Dynamics license is required to use the Detailed Heat Loss model found on this page.

4.4.5 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the Heat Exchanger unit operation.

To view the stream parameters broken down per stream phase, open the Worksheet tab of the stream property view.

The PF Specs page is relevant to dynamics cases only.

4.4.6 Performance Tab

The Performance tab has pages that display the results of the Heat Exchanger calculations in overall performance parameters, as well as using plots and tables.

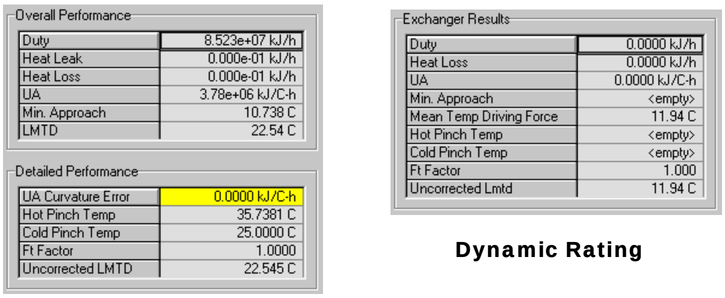
The Performance tab contains the following pages:

- Details
- Plots
- Tables
- Setup
- Error Msg

Details Page

The information from the Details page appears in the figure below.

Figure 4.75



Overall Performance

Duty	8.523e+07 kJ/h
Heat Leak	0.000e-01 kJ/h
Heat Loss	0.000e-01 kJ/h
UA	3.78e+06 kJ/C-h
Min. Approach	10.738 C
LMTD	22.54 C

Detailed Performance

UA Curvature Error	0.0000 kJ/C-h
Hot Pinch Temp	35.7381 C
Cold Pinch Temp	25.0000 C
Ft Factor	1.0000
Uncorrected LMTD	22.545 C

Exchanger Results

Duty	0.0000 kJ/h
Heat Loss	0.0000 kJ/h
UA	0.0000 kJ/C-h
Min. Approach	<empty>
Mean Temp Driving Force	11.94 C
Hot Pinch Temp	<empty>
Cold Pinch Temp	<empty>
Ft Factor	1.000
Uncorrected Lmtd	11.94 C

Steady State Rating

Dynamic Rating

The appearance of this page is slightly different for the Dynamic Rating model.

Overall Performance Group

The Overall and Detailed performance groups contain the following parameters that are calculated by HYSYS:

Parameter	Description
Duty	Heat flow from the hot stream to the cold stream.
Heat Leak	Loss of cold side duty due to leakage. Duty gained to reflect the increase in temperature.
Heat Loss	Loss of the hot side duty to leakage. The overall duty plus the heat loss is equal to the individual hot stream duty defined on the Tables page.
UA	Product of the Overall Heat Transfer Coefficient, and the Total Area available for heat transfer. The UA is equal to the overall duty divided by the LMTD.
Minimum Approach	The minimum temperature difference between the hot and cold stream.
Mean Temp Driving Force	The average temperature difference between the hot and cold stream.

Parameter	Description
LMTD	The uncorrected LMTD multiplied by the Ft factor. For the Weighted Rating Method, the uncorrected LMTD equals the effective LMTD.
UA Curvature Error	The LMTD is ordinarily calculated using constant heat capacity. An LMTD can also be calculated using linear heat capacity. In either case, a different UA is predicted. The UA Curvature Error reflects the difference between these UAs.
Hot Pinch Temperature	The hot stream temperature at the minimum approach.
Cold Pinch Temperature	The cold stream temperature at the minimum approach.
F_t Factor	The LMTD (log mean temperature difference) correction factor, F_t, is calculated as a function of the Number of Shell Passes and the temperature approaches. For a counter-current Heat Exchanger, F_t is 1.0. For the Weighted rating method, F_t = 1.
Uncorrected LMTD	(Applicable only for the End Point method) - The LMTD is calculated in terms of the temperature approaches (terminal temperature differences) in the exchanger, using the **Equation (4.35)**.

Uncorrected LMTD equation:

$$\Delta T_{LM} = \frac{\Delta T_1 - \Delta T_2}{\ln(\Delta T_1 / (\Delta T_2))} \tag{4.35}$$

where:

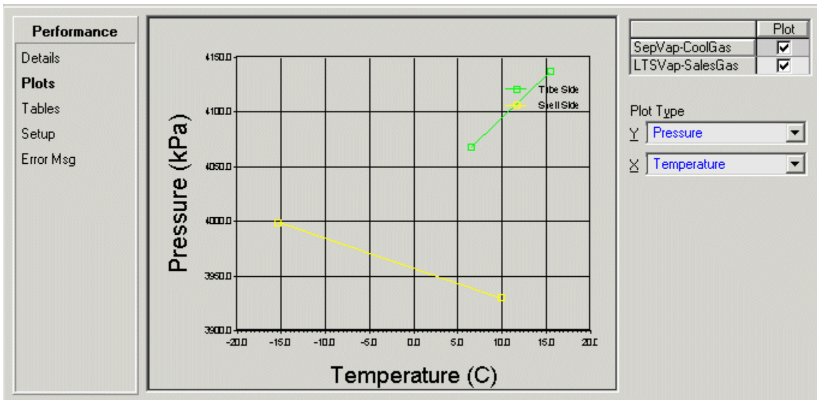
$$\Delta T_1 = T_{hot,\ out} - T_{cold,\ in}$$

$$\Delta T_2 = T_{hot,\ in} - T_{cold,\ out}$$

Plots Page

You can plot curves for the hot and/or cold fluid. Use the Plot checkboxes to specify which side(s) of the exchanger should be plotted.

Figure 4.76



Refer to **Section 1.3.1 - Graph Control Property View** for more information.

You can modify the appearance of the plot via the Graph Control property view.

The following default variables can be plotted along either the X or Y-axis:

- Temperature
- UA
- Delta T
- Enthalpy
- Pressure
- Heat Flow

Select the combination from the Plot Type drop-down list. To Plot other available variables, you need to add them on the Setup page. Once the variables are added, they are available in the X and Y drop-down lists.

Tables Page

On the Tables page, you can view (default variables) interval temperature, pressure, heat flow, enthalpy, UA, and vapour fraction for each side of the Exchanger in a tabular format. Select either the Shell Side or Tube Side radio button.

To view other available variables, you need to add them on the Setup page. Variables are displayed based on Phase Viewing Options selected.

Setup Page

The Setup page allows you to filter and add variables to be viewed on the Plots and Tables pages.

The variables that are listed in the Selected Viewing Variables group are available in the X and Y drop down list for plotting on the Plots page. The variables are also available for tabular plot results on the Tables page based on the Phase Viewing Options selected.

Error Msg Page

The Error Msg page contains a list of the warning messages on the Heat Exchanger. You cannot add comments to this page. Use it to see if there are any warnings in modeling the Heat Exchanger.

4.4.7 Dynamics Tab

The Dynamics tab contains the following pages:

- Model
- Specs
- Holdup
- Stripchart

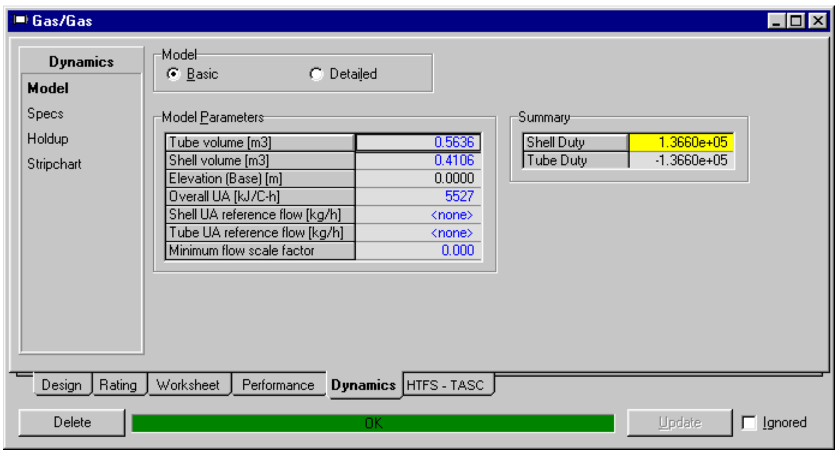
If you are working exclusively in Steady State mode, you are not required to change any information on the pages accessible through the Dynamics tab.

Any information specified on the Rating tab also appears in the Dynamics tab.

Model Page

In the Model page, you can specify whether HYSYS uses a Basic or Detailed model.

Figure 4.77



Basic Model

The Model Parameters group contains the following information for the Heat Exchanger unit operation:

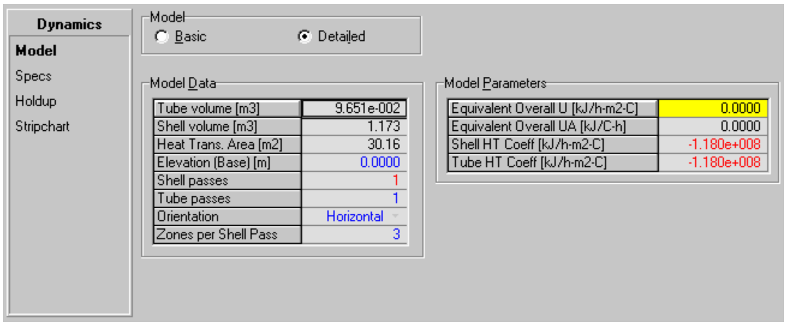
Field	Description
Tube/Shell Volume	The volume of the shell and tube must be specified in the Basic model.
Elevation	The elevation is significant in the calculation of static head around and in the Heat Exchanger.
Overall UA	Product of the Overall Heat Transfer Coefficient and the Total Area available for heat transfer. The Heat Exchanger duty is proportional to the log mean temperature difference, where UA is the proportionality factor. The UA must be specified if the Basic model is used.
Shell/Tube UA Reference Flow	Since UA depends on flow, these parameters allow you to set a reference point that uses HYSYS to calculate a more realistic UA value. If no reference point is set then UA is fixed. If the UA is specified, the specified UA value does not change during the simulation. The UA value that is used, however, does change if a Reference Flow is specified. Basically, as in most heat transfer correlation's, the heat transfer coefficient is proportional to the $(\text{mass flow ratio})^{0.8}$. The equation below is used to determine the UA used: $$UA_{\text{used}} = UA_{\text{specified}} \times \left(\frac{\text{mass flow}_{\text{current}}}{\text{mass flow}_{\text{reference}}}\right)^{0.8} \quad (4.36)$$ Reference flows generally help to stabilize the system when you do shut downs and startups as well.
Minimum Flow Scale Factor	The ratio of mass flow at time *t* to reference mass flow is also known as flow scaled factor. The minimum flow scaled factor is the lowest value which the ratio is anticipated at low flow regions. This value can be expressed in a positive value or negative value. • A positive value ensures that some heat transfer still takes place at very low flows. • A negative value ignores heat transfer at very low flows. A negative factor is often used in shut downs if you are not interested in the results or run into problems shutting down an exchanger. If the Minimum Flow Scale Factor is specified, the **Equations (4.36)** uses the $\left(\frac{\text{mass flow}_{\text{current}}}{\text{mass flow}_{\text{reference}}}\right)^{0.8}$ ratio if the ratio is greater than the Min Flow Scale Factor. Otherwise the Min Flow Scale Factor is used. In some cases you can use a negative value for minimum flow scale factor. If you use -0.1, then if the scale factor goes below 0.1, the Minimum Flow Scale Factor uses 0.

The Summary group contains information regarding the duty of the Heat Exchanger shell and tube sides.

Detailed Model

When you select the Detailed radio button, a summary of the rating information specified on the Rating tab appears.

Figure 4.78



The Model Data group contains the following information:

Field	Description
Tube/Shell Volume	The volume of the shell and tube is calculated from the Heat Exchanger rating information.
Heat Transfer Area	The heat transfer area is calculated from the Heat Exchanger rating information.
Elevation	The elevation is significant in the calculation of static head around and in the Heat Exchanger.
Shell/Tube Passes	You can specify the number of tube and shell passes in the shell of the Heat Exchanger. In general, at least 2n tube passes must be specified for every n shell pass. The exception is a counter-current flow Heat Exchanger which has 1 shell pass and one tube pass
Orientation	The orientation may be specified as a vertical or horizontal Heat Exchanger. The orientation of the Heat Exchanger does not impact the steady state solver. However, it used in the dynamic Heat Exchanger in the calculation of liquid level in the shell.
Zones per Shell Pass	Enter the number of zones you would like for one shell pass. The total number of zones in a Heat Exchanger shell is calculated as: $$Total\ Zones = \#\text{ of }Shells \cdot \frac{Zones}{Shell\ Pass}$$

The Model Parameters group contains the local and overall heat transfer coefficients for the Heat Exchanger. Depending on how the Heat Transfer Coefficient Calculator is set on the Parameters page of the Rating tab, the local and overall heat transfer coefficients can either be calculated or specified in the Model Parameters group.

HT Coefficient Calculator Setting	Description
Shell & Tube	Overall heat transfer coefficient, U, is calculated using the exchanger rating information.
U Specified	Overall heat transfer coefficient, U, is specified by you.

The Startup Level group appears only if the Heat Exchanger is specified with a single shell and/or tube pass having only one zone. The Startup level cannot be set for multiple shell and/or tube pass exchangers for multiple shell or tube passes. You can specify an initial liquid level percent for the shell or tube holdups. This initial liquid level percent is used only if the simulation case re-initializes.

Specs Page

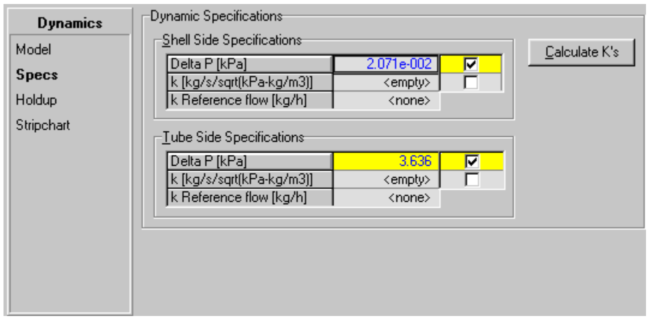
The Specs page contains information regarding the calculation of pressure drop across the Heat Exchanger.

The information displayed on the Specs page depends on the model (Basic or Detailed) selected on the Model page.

Basic Model

When you select the Basic model radio button on the Model page, the Specs page appears as follows.

Figure 4.79



The pressure drop across any pass in the Heat Exchanger operation can be determined in one of two ways:

- Specify the pressure drop.
- Define a pressure flow relation for each pass by specifying a k value.

The following parameters are used to specify the pressure drop for the Heat Exchanger.

Dynamic Specification	Description
Shell/Tube Delta P	The pressure drop across the Shell/Tube side of the Heat Exchanger may be specified (checkbox active) of calculated (checkbox inactive).
k	Activate this option if to have the Pressure Flow k values used in the calculation of pressure drop.

Dynamic Specification	Description
k Reference Flow	If the pressure flow option is chosen the k value is calculated based on two criteria. If the flow of the system is larger than the k Reference Flow, the k value remains unchanged. If the flow of the system is smaller than the k Reference Flow the k value is given by: $$k_{used} = k_{specified} \times Factor$$ *where:* *Factor = value is determined by HYSYS internally to take into consideration the flow and pressure drop relationship at low flow regions.* At low flow range, it is recommended that the k reference flow is taken as 40% of steady state design flow for better pressure flow stability.

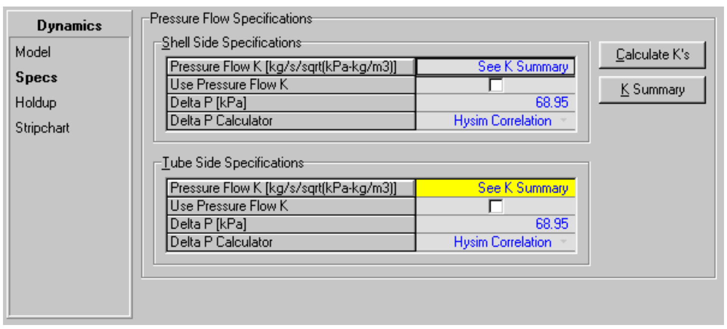
Effectively, the k Reference Flow results in a more linear relationship between flow and pressure drop, and this is used to increase model stability during startup and shutdown where the flows are low.

Use the Calculate k button to calculate a k value based on the Delta P and k Reference flow. Ensure that there is a non zero pressure drop across the Heat Exchanger before you click the Calculate k button.

Detailed Model

When you select the Basic model radio button on the Model page, the Specs page appears as follows.

Figure 4.80



The following parameters are used to specify the pressure drop for the Heat Exchanger.

Dynamic Specification	Description
Pressure Flow k	The k-value defines the relationship between the flow through the shell or tube holdup and the pressure of the surrounding streams. You can either specify the k-value or have it calculated from the stream conditions surrounding the Heat Exchanger. you can “size” the exchanger with a k-value by clicking the **Calculate K’s** button. Ensure that there is a non zero pressure drop across the Heat Exchanger before the Calculate k button is clicked.
Pressure Flow Option	Activate this option to have the Pressure Flow k values used in the calculation of pressure drop. If the Pressure Flow option is selected, the Shell/Tube Delta P calculator must also be set to non specified.
Shell/Tube Delta P	The pressure drop across the Shell/Tube side of the Heat Exchanger may be specified or calculated.
Shell/Tube Delta P Calculator	The Shell/Tube Delta P calculator allows you to either specify or calculate the shell/tube pressure drop across the Heat Exchanger. Specify the cell with one of the following options: • **Shell & Tube Delta P Calculator**. The pressure drop is calculated using the Heat Exchanger rating information and correlations. • **user specified**. The pressure drop is specified by you. • **not specified**. This option is only applicable in Dynamic mode. Pressure drop across the Heat Exchanger is calculated from a pressure flow relationship. You must specify a k-value and activate the Pressure Flow option to use this calculator.

Refer to **Detailed Heat Model Properties** section for more information.

Clicking the **K Summary** button opens the Detailed Heat Model Properties property view.

Holdup Page

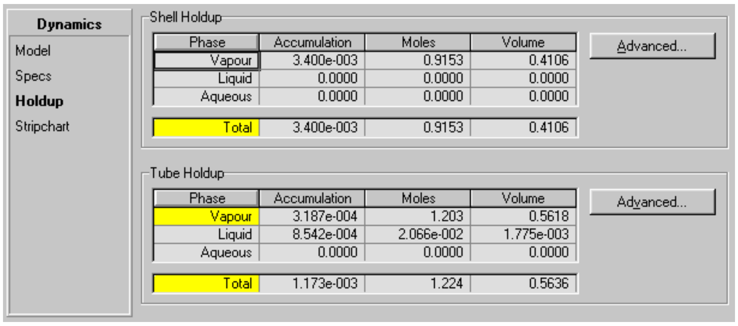
Refer to **Section 1.3.3 - Holdup Page** for more information.

The Holdup page contains information regarding the shell and tube holdup properties, composition, and amount.

Basic Model

When you select the Basic model radio button on the Model page, the Holdup page appears as follows.

Figure 4.81

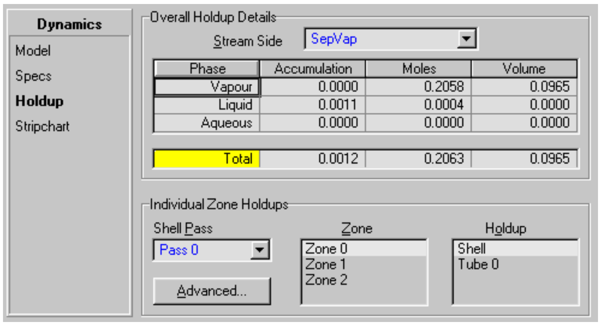


The Shell Holdup group and Tube Holdup group contain information regarding the shell and tube side holdup parameters.

Detailed Model

When you select the Detailed model radio button on the Model page, the Holdup page appears as follows.

Figure 4.82



The Overall Holdup Details group contains information regarding the shell and tube side holdup parameters.

The Individual Zone Holdups group contains detailed holdup properties for every layer in each zone of the Heat Exchanger unit operation. In order to view the advanced properties for individual holdups, you must first choose the individual holdup.

To choose individual holdups you must specify the Zone and Layer in the corresponding drop-down lists.

Stripchart Page

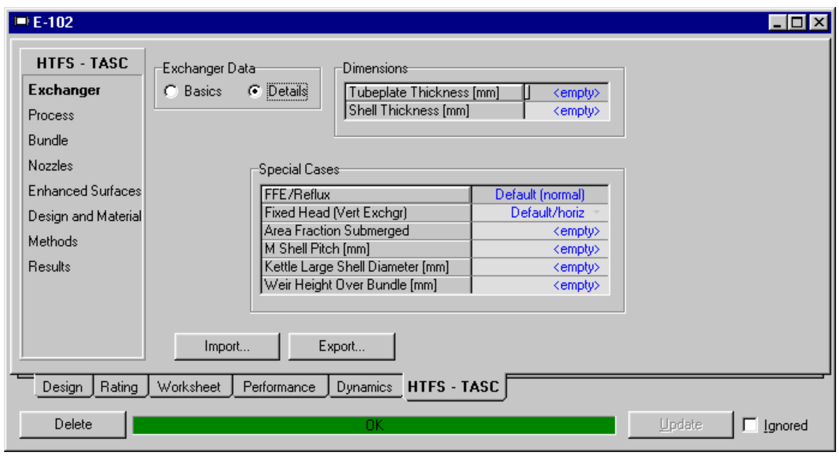
Refer to **Section 1.3.7 - Stripchart Page / Tab** for more information.

The Stripchart page allows you to select and create default strip charts containing various variable associated to the operation.

4.4.8 HTFS-TASC Tab

When you select the HTFS - Engine model on the Parameters page of the Design tab, the HTFS-TASC tab appears as shown in the figure below:

Figure 4.83



The HTFS-TASC tab contains the following pages:

- Exchanger
- Process
- Bundle
- Nozzles
- Enhanced Surfaces
- Design and Material
- Methods
- Results

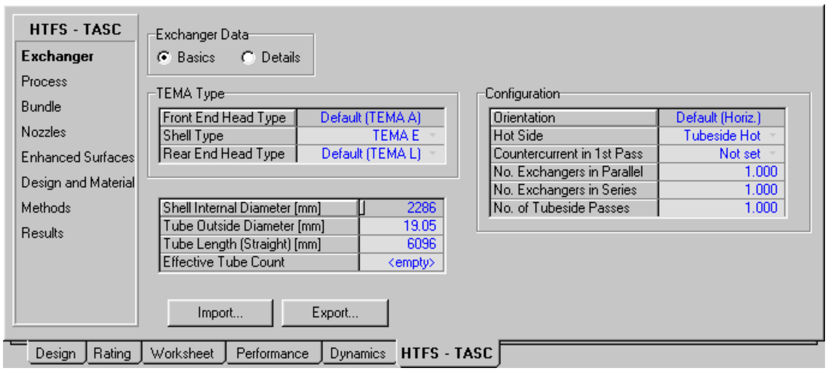
The HTFS-TASC tab also contains two buttons:

- **Import**. Allows you to import values from TASC into the pages of the tab.
- **Export**. Allows you to export the information provided within this tab to TASC.

Exchanger Page

The Exchanger page allows you to input parameters that define the geometric configuration of the Heat Exchanger.

Figure 4.84



After entering a basic configuration of the Heat Exchanger, you can specify detailed information.

Basics Data

For the Basics data, you can enter the following information:

Refer to the **TASC Thermal Reference** guide for more information about the selections available.

Entry	Description
Front End Head Type	You can select the type of front end head for your heat exchanger using the drop-down list. The type of head selected has no significant effect on the heat exchanger thermal or pressure drop performance, as calculated by TASC. It only affects the heat exchanger weight.
Shell Type	You can select the type of shells for the heat exchanger using the drop-down list.
Rear End Head Type	You can select the type of rear end head for your heat exchanger using the drop-down list.
Shell Internal Diameter	You can enter the internal diameter of the shell in this cell.
Tube Outside Diameter	You can enter the outside diameter of the tube in this cell.
Tube Length (Straight)	You can enter the length of the tube in this cell.
Effective Tube Count	You can enter the number of tubes in the heat exchanger in this cell. If you did not enter any value in this cell, TASC derives an exact tube count while setting up the Tube Bundle Layout.
Orientation	You can select from three types of orientation for your heat exchanger in the drop-down list: • Default (Horiz.) • Horizontal • Vertical
Hot Side	You can select which side is the hot side in your heat exchanger from the drop-down list. There are three selections: • Not yet set • Tubeside hot • Shell-side hot
Countercurrent in 1st Pass	You can select whether countercurrent occurs in the first pass from the drop-down list. There are three selections: • Not set • Yes • No (co-current)
No. Exchangers in Parallel	You can specify how many heat exchangers are parallel to the current heat exchanger in this cell.
No. Exchangers in Series	You can specify how many heat exchangers are in series to the current heat exchanger in this cell.
No. of Tubeside Passes	You can specify how many tubeside passes occur in the heat exchanger in this cell.

Details Data

For the Details data, you can enter the following information:

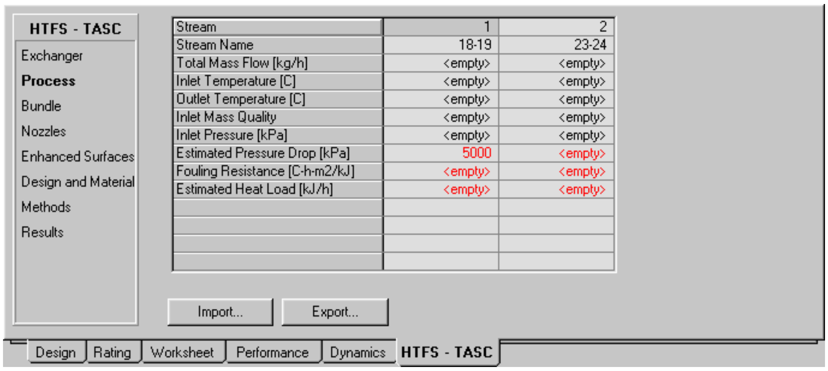
Refer to the **TASC Thermal Reference** guide for information about the selections available.

Entry	Description
Tubeplate Thickness	You can specify the tubeplate thickness in this cell.
Shell Thickness	You can specify the shell thickness in this cell.
FFE/Reflux	You can select the special type of exchanger using the drop-down list. There are four selections: • Default (normal) • Normal exchanger • Falling Film Evap • Reflux Condenser
Fixed Head (Vert Exchgr)	You can select the location of the fixed end head from the drop-down list. There are three selections: • Default/horiz • Top • Bottom The Top and Bottom selections only apply to vertical shells.
Area Fraction Submerged	You can enter the area fraction on the tubes that may be submerged under condensate in this cell. This value only applies to horizontal shellside condensers and if there is a lute or geometric feature that causes tubes to be submerged.
M Shell Pitch	You can enter the shell pitch for double-pipe U-tube exchangers or Multitube hairpin exchangers in this cell. The value is used to determine the U-bend heat transfer area.
Kettle Large Shell Diameter	You can enter the internal diameter of the larger part of the shell of a kettle reboiler in this cell.
Weir Height Over Bundle	You can enter the height of the weir above the top of the bundle in this cell. This value is used to define the head of liquid providing the driving force for re-circulation within a kettle. If no value is entered, HYSYS assumes the value is zero. The top of the weir is assumed to be level with the top of the outer tube limit circle of the bundle.

Process Page

The Process page allows you to specify the estimate pressure drop, fouling resistance, and heat load.

Figure 4.85



The estimated heat load is used as a starting point to do the simulation calculation.

Bundle Page

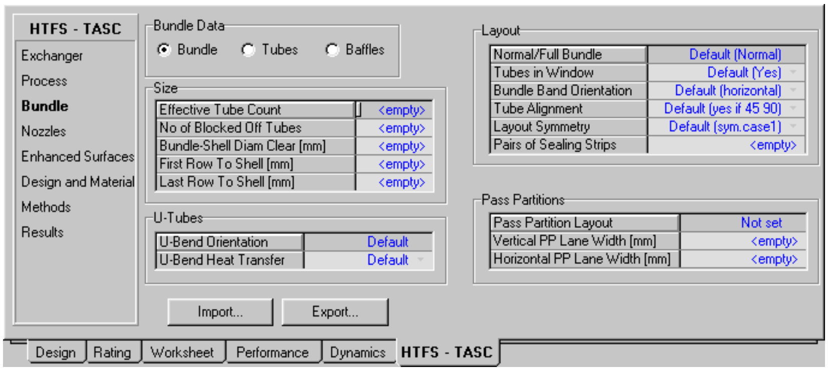
The Bundle page allows you to specify the bundle, tube, and baffles configurations. The radio buttons in the Bundle Data group controls which configuration appears on the page.

- Bundle
- Tubes
- Baffles

Bundle Configuration

If you select the Bundle radio button in the Bundle Data group, the Bundle page appears as shown in the figure below:

Figure 4.86



The configuration information you can specify for the bundle is sorted into four groups:

- Size
- U-Tubes
- Layout
- Pass Partitions

Size Group

The Size group allows you to specify information used to calculate the size of the bundle.

Specification	Description
Effective Tube Count	Number of tubes in the heat exchanger. The Effective Tube Count field is linked to the Effective Tube Count field on the Exchanger page. Any changes in either fields propagates to the other.
No of Blocked Off Tubes	Number of blocked off tubes.
Bundle-Shell DIam Clear	Diametral clearance between the tube bundle (outer limit diameter) and the shell wall. This value is used to determine the fraction of the shellside flow which by passes around the bundle. For zero clearance, enter **0**.

Specification	Description
First Row to Shell	Specify the distance between the centres of the first row tubes to the shell. The first tube row is that nearest the inlet nozzle.
Last Row to Shell	Specify the distance between the centres of the last row tubes to the shell. The last tube row is that furthest from the inlet nozzle.

U-Tubes Group

The U-tubes group allows you to select the configuration of the U-tubes.

Refer to the **TASC Thermal Reference** guide for information about the selections available.

Specification	Description
U-Bend Orientation	You can select the type of U-bend orientation from the drop-down list. There are three selections: • Default • Horizontal • Vertical
U-Bend Heat Transfer	You can select whether to include or exclude the heat transfer that occurs in the U-tube using the drop-down list. There are three selections: • Default • Allow for U-bend • Ignore U-bend

Layout Group

The Layout group allows you to specify information used to design the layout of the bundle.

Refer to **TASC Thermal Reference** guide for information about the selections available.

Specification	Description
Normal/Full Bundle	You can select what type of bundle to use from the drop-down list. There are three selections: • Default (Normal) • Normal Bundle • Full Bundle
Tubes in Window	You can select whether you want tubes in the window or not from the drop-down list. There are three selections: • Default (Yes) • Yes • No

Specification	Description
Bundle Band Orientation	You can select the bundle band orientation from the drop-down list. There are three selections: • Default (horizontal) • Horizontal • Vertical
Tube Alignment	You can select the tube alignment from the drop-down list. There are four selections: • Default (if yes 45 90) • Fully aligned • Unaligned • Part aligned
Layout Symmetry	You can select the layout symmetry from the drop-down list. There are four selections: • Default (sym.case 1) • Symmetry (case 1) • Symmetry (case 2) • Not enforced
Pairs of Sealing Strips	Number of pairs of sealing strips.

Pass Partitions Group

The Pass Partitions group allows you to specify information used to configure the pass partition.

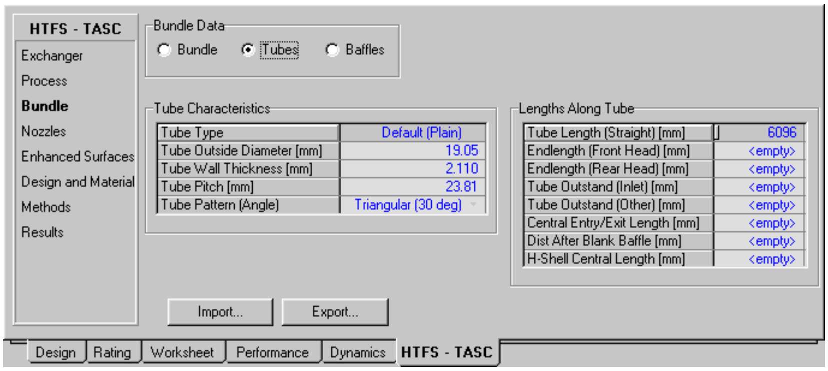
Refer to the **TASC Thermal Reference** guide for information about the selections available.

Specification	Description
Pass Partition Layout	You can select the type of pass partition from the drop-down list. There are four selections: • Not set • H Banded • Double Banded • Ribbon Banded
Vertical PP Lane Width	Vertical pass partition lane width.
Horizontal PP Lane Width	Horizontal pass partition lane width.

Tubes Configuration

If you select the Tubes radio button in the Bundle Data group, the Bundle page appears as shown in the figure below:

Figure 4.87



The configuration information you can specify for the tubes is sorted into two groups:

- Tube Characteristics
- Lengths Along Tube

Tube Characteristics Group

The Tube Characteristics group allows you to specify the configuration for the tube.

Refer to the **TASC Thermal Reference** guide for information about the selections available.

Specification	Description
Tube Type	You can select the type of tube you want from the drop-down lists: • Default (Plain) • Plain Tubes • Lowfin Tubes • Longitudinal Tubes
Tube Outside Diameter	Outside diameter of the tube.
Tube Wall Thickness	Thickness of the tube's wall.

Specification	Description
Tube Pitch	The tube's pitch.
Tube Pattern (Angle)	You can select the pattern of the tube from the drop-down list: • Default (Triangular) • Triangular (30 deg) • Rotated square (45) • Roated triang. (60) • Square (90 deg)

Lengths Along Tube Group

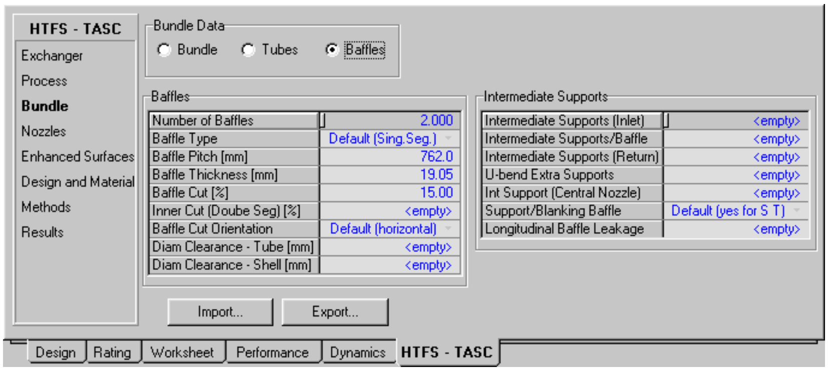
The Lengths Along Tube group allows you to specify the lengths of each tube section.

Specification	Description
Tube Length	Length of the tube.
Endlength (Front Head)	Length of the front head of the tube.
Endlength (Rear Head)	Length of the rear head of the tube.
Tube Outstand (Inlet)	The distance the tube inlet end protrudes beyond the face of a tube sheet.
Tube Outstand (Other)	The distance the tube rear end protrudes beyond the face of a tube sheet.
Central Entry / Exit Length	The distance between the centres of the Flow Baffles on either side of a central inlet or outlet nozzle. HYSYS assumes the two baffle spacings are equal if no value is entered.
Dist. After Blank Baffle	The distance between the tube and the blank baffle.
H-Shell Central Length	Length of the central region in an H-shell. This value is the distance between two halves of the axial baffle in an H-shell. HYSYS assumes the value to be double the mean length of the end spaces at the ends of the exchanger if no value is entered.

Baffles Configuration

If you select the Baffles radio button in the Bundle Data group, the Bundle page appears as shown in the figure below:

Figure 4.88



The configuration information you can specify for the baffles is sorted into two groups:

- Baffles
- Intermediate Supports

Baffles Group

The Baffles group allows you to specify the configuration of the baffles.

Refer to the **TASC Thermal Reference** guide for information about the selections available.

Specification	Description
Number of Baffles	Number of baffles.
Baffle Type	Select the baffle type from the drop-down list: • Default (Sing.Seg.) • Single Segmental • Double Segmental • Unbar/Low pr.drop • Rodbaffled
Baffle Pitch	The value of the baffle pitch. The baffle pitch is the baffle spacing plus the baffle thickness.
Baffle Thickness	The baffle thickness.
Baffle Cut	The percentage of baffle cut.

Specification	Description
Inner Cut (Double Seg)	The percentage of inner cut. This is only applicable to Double Segmental baffle type.
Baffle Cut Orientation	Select the orientation of the baffle cut using the drop-down list: • Default (horizontal) • Vertical • Horizontal
Diam. Clearance - Tube	Diametral clearance between the tube and the baffle hole. For a zero clearance, enter **0**.
Diam. Clearance - Shell	Diametral clearance between the baffles and the shell wall. For a zero clearance, enter **0**.

Intermediate Support Group

The Intermediate Support group allows you to specify the tube supports, other than flow baffles, that help remove the risk of vibration damage.

Specification	Description
Intermediate Supports (Inlet)	Number of intermediate supports in the inlet endspace. This endspace corresponds to the inlet endlength.
Intermediate Supports/Baffle	Number of intermediate supports between each pair of flow baffles.
Intermediate Supports (Return)	Number of intermediate supports in the endspace corresponding to the outlet (return) endlength.
U-bend Extra Supports	Number of tube supports on the U-bend.
Int. Supports (Central Nozzle)	Number of intermediate supports for nozzles (not over inlet or return endspace).
Support/Blanking Baffle	Select whether there is a support of blanking baffle at the rear end head: • Default (Yes for S T) • Yes • No
Longitudinal Baffle Leakage	An estimate of the percentage of the shellside flow which leaks across the longitudinal baffle. This value is only relevant to the F, G, or H shell types.

Refer to the **TASC Thermal Reference** guide for information about the selections available.

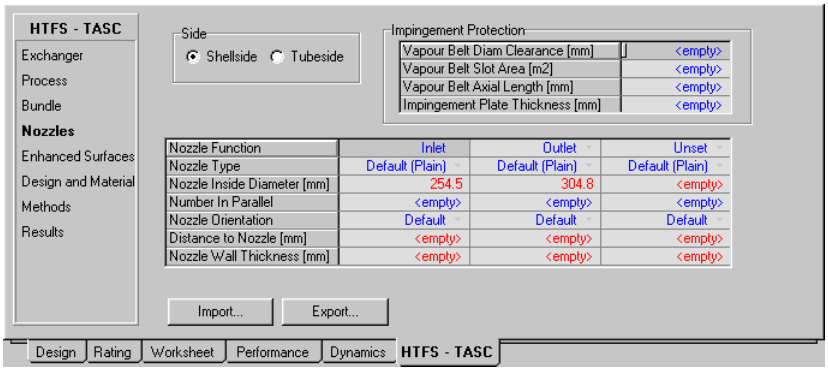
Nozzles Page

The Nozzles page allows you to specify the nozzles in the shellside and tubeside. The radio buttons in the Side Data group controls which side appears on the page.

Shellside Configuration

If you select the Shellside radio button in the Size group, the Nozzles page appears as shown in the figure below:

Figure 4.89



The following table lists and describes the configuration information that you can specify for the nozzles in shellside.

Specification	Description
Vapour Belt Diam Clearance	Diametral annular clearance (difference in diameters) between the outside of the shell and the vapour belt.
Vapour Belt Slot Area	The total flow area of all the slots leading through the shell wall (from the vapour belt into the shell).
Vapour Belt Axial Length	The axial length of the exchanger occupied by (the inside of) the belt.
Impingement Plate Thickness	The thickness of the impingement plate.
Nozzle Function	You can specify up to three types of nozzle function. Select the nozzle function from the drop-down list: • Unset • Inlet • Outlet • Intermediate • Liquid Outlet • Vapour Outlet

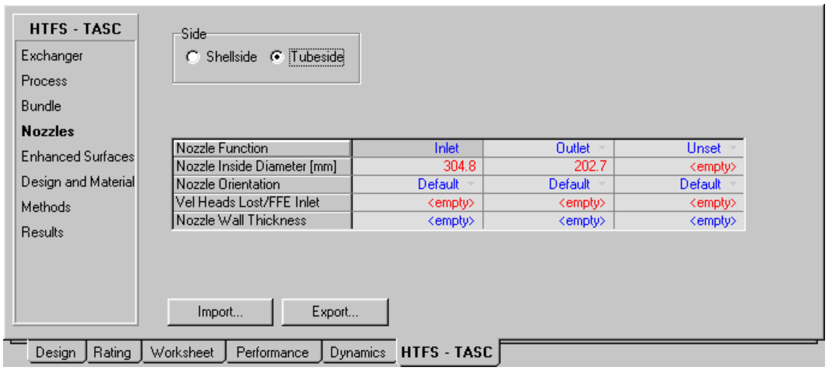
Refer to the **TASC Thermal Reference** guide for information about the selections available.

Specification	Description
Nozzle Type	Select the nozzle types from the drop-down list: • Default (Plain) • Plain • Plain + Imp Plate • Vapour Belt
Nozzle Inside Diameter	The inside diameter of the nozzle.
Number In Parallel	Number of nozzles in parallel on one shell.
Nozzle Orientation	Select the nozzle orientation from the drop-down list: • Default • Top of Shell • RHSide of Shell • Bottom of Shell • LHSide of Shell
Distance to Nozzle	The axial distance along the shell to the nozzle centre line, measured from the inner surface of the tubesheet at the front (fixed) head.
Nozzle Wall Thickness	The wall thickness of the nozzle.

Tubeside Configuration

If you select the Tubeside radio button in the Size group, the Nozzles page appears as shown in the figure below:

Figure 4.90



The configuration information you can specify for the nozzles in tubeside is described in the table below:

Refer to the **TASC Thermal Reference** guide for information about the selections available.

Specification	Description
Nozzle Function	You can specify up to three types of nozzle function. Select the nozzle function from the drop-down list: • Unset • Inlet • Outlet • Intermediate • Liquid Outlet • Vapour Outlet
Nozzle Inside Diameter	The inside diameter of the nozzle.
Nozzle Orientation	Select the nozzle orientation from the drop-down list: • Default • Top of Shell • RHSide of Shell • Bottom of Shell • LHSide of Shell
Vel Head Lost / FFE Inlet	Number of velocity heads lost in a device (used to achieve uniform flow distribution of the liquid in-flow to all the tubes of a falling film evaporator).
Nozzle Wall Thickness	The wall thickness of the nozzle.

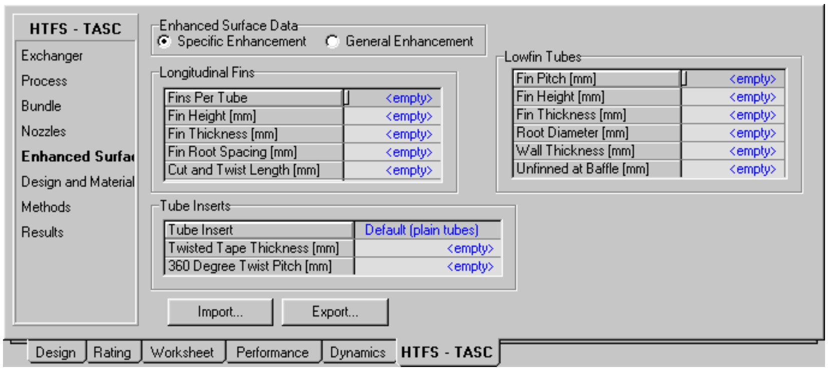
Enhanced Surface Page

The Enhanced Surface page allows you to perform model calculations on the exchanger that are not explicitly modeled by TASC. There are two enhanced options on the page, and you can select which enhanced option you want using the radio buttons in the Enhanced Surface Data group.

Specific Enhanced Option

If you select the Specific Enhanced radio button in the Enhanced Surface Data group, the Enhanced Surface page appears as shown in the figure below.

Figure 4.91



The variables you can specify for the Specific Enhanced option are sorted into three groups:

- Longitudinal Fins
- Lowfin Tubes
- Tube Inserts

Longitudinal Fins Group

The Longitudinal Fins group allows you to specify the configuration of the longitudinal fins.

Specification	Description
Fins Per Tube	Number of fins are on each tube.
Fin Height	Height of each fin.
Fin Thickness	Thickness of each fin.
Fin Root Spacing	The root spacing of each fin.
Cut and Twist Length	The cut and twist length.

Lowfin Tubes Group

The Lowfin Tubes group allows you to specify the configuration of the lowfin tubes.

Specification	Description
Fin Pitch	The lowfin fin pitch.
Fin Height	The height of each fin.
Fin Thickness	The thickness of each fin.
Root Diameter	The lowfin tube root diameter.
Wall Thickness	The lowfin tube wall thickness.
Unfinned at Baffle	Length of unfinned tubing at a baffle.

Tube Inserts Group

The Tube Inserts group allows you to specify the configuration of the tube inserts.

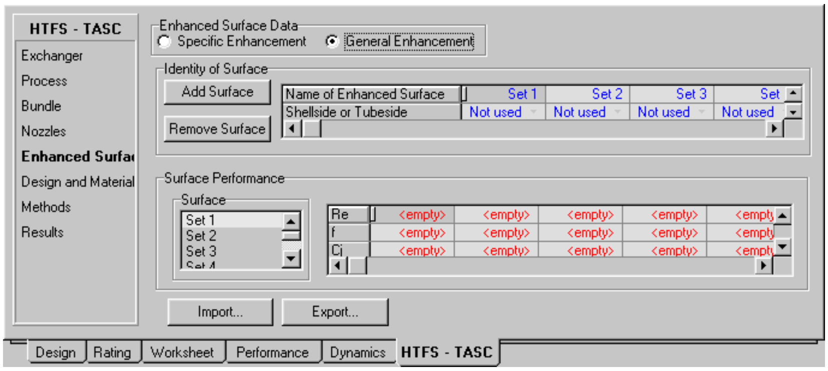
Refer to the **TASC Thermal Reference** guide for information about the selections available.

Specification	Description
Tube Insert	Select the type of tube inserts from the drop-down list: • Default (plain tubes) • None (plain tubes) • Twisted tape
Twisted Tape Thickness	The twisted tape thickness. The value only applies if you selected twisted tape for the tube insert.
360 Degree Twisted Pitch	The distance between each 360 degree twist of a twisted tape insert.

Specific Enhanced Option

If you select the General Enhanced radio button in the Enhanced Surface Data group, the Enhanced Surface page appears as shown in the figure below:

Figure 4.92



The variables you can specify for the General Enhanced option is sorted into two groups:

- Identity of Surface
- Surface Performance

Identity of Surface Group

The Identity of Surface group allows you to create surfaces for both the shellside and tubeside.

Specification	Description
Add Surface	Allows you to add/create a surface.
Remove Surface	Allows you to remove the last surface.
Name of Enhanced Surface	Contains the name of the surface created. HYSYS automatically names the surface as “Set” followed by a number. The number value is incremented by 1 for each new surface created.
Shellside or Tubeside	Select which side the surface created on from the drop-down list: • Not used • Shellside • Tubeside

Refer to the **TASC Thermal Reference** guide for information about the selections available.

Surface Performance Group

The Surface Performance group allows you to specify the configuration of each surface.

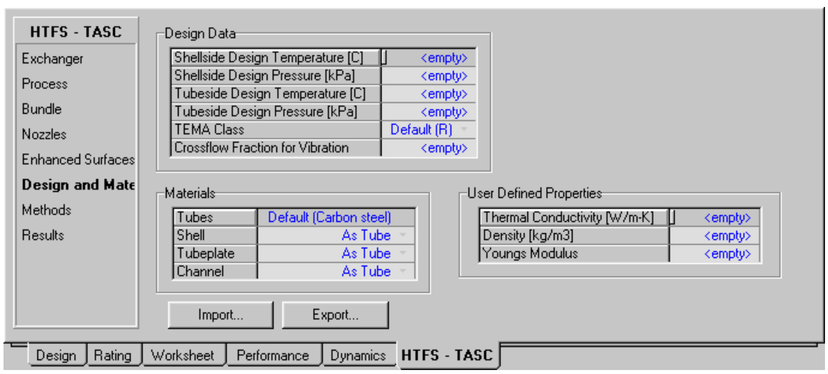
Specification	Description
Surface	Contains the list of surfaces created. Any values entered in the table located at the right of the list apply only to the surface you selected in the list.
Re	The Reynolds Number for the corresponding surface.
f	The friction factor for the corresponding surface.
Cj	The heat transfer factor (Colburn j factor) for the corresponding surface.

Design and Material Page

The Design and Material page allows you to specify design values, material types, and some properties for the Heat Exchanger. The information on this page is sorted into three groups:

- Design Data
- Materials
- User Defined Properties

Figure 4.93



Design Data Group

The Design Data group allows you to specify the following variables:

Refer to the **TASC Thermal Reference** guide for information about the selections available.

Specification	Description
Shellside Design Temperature	Design temperature on the shellside.
Shellside Design Pressure	Design pressure on the shellside.
Tubeside Design Temperature	Design temperature on the tubeside.
Tubeside Design Pressure	Design pressure on the tubeside.
TEMA Class	Select the TEMA class from the drop-down list: • Default (R) • R • C • B • Not TEMA
Crossflow Fraction for Vibration	The fraction from the shellside flow in the cross flow which causes vibration.

Materials Group

The Materials group allows you to select the material type for the heat exchanger. HYSYS lets you select the material for four parts of the heat exchanger: Tubes, Shell, Tubeplate, and Channel. You can select the material type from the drop-down list provided for each part.

User Defined Properties Group

The User Defined Properties group allows you to specify values for the following properties:

Specification	Description
Thermal Conductivity	The thermal conductivity of the tube material. This value overrides the calculated value based on the tube material selected.
Density	Density for all the exchanger materials. This value overrides the calculated value based on the selected materials for each part of the exchanger.
Youngs Modulus	The Young's Modulus. This value overrides the calculated value based on the tube material selected.

Methods Page

The Methods page allows you to specify the process methods and constraints of the heat exchanger. The Methods and Constraints group contains three radio buttons:

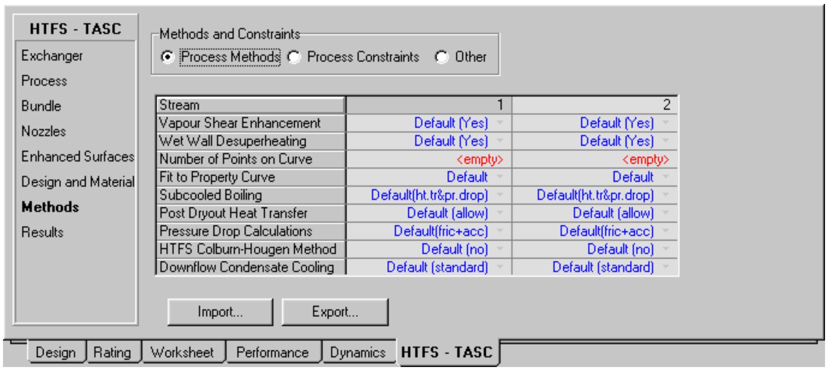
- Process Methods
- Process Constraints
- Other

The variables displayed on this page depend on the radio button you selected in the Methods and Constraints group.

Process Methods Variables

If you select the Process Methods radio button from the Methods and Constraints group, the Methods page appears as shown in the figure below:

Figure 4.94



The table below lists the variables available for the process method:

Refer to the **TASC Thermal Reference** guide for information about the selections available.

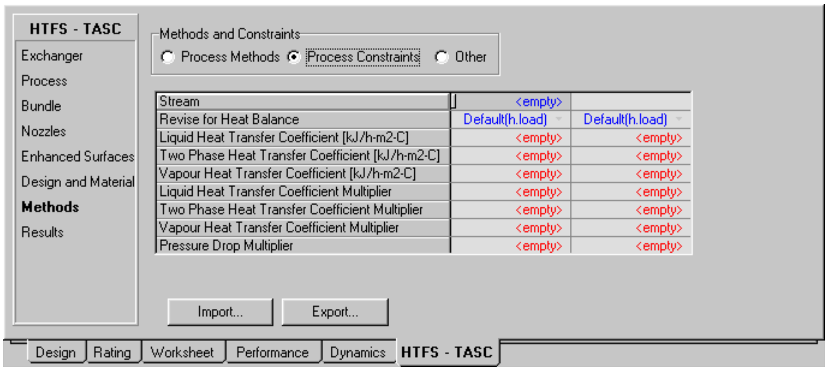
Method	Description
Vapour Shear Enhancement	Select whether the process stream has vapour shear enhancement from the drop-down list: • Default (Yes) • Yes • No
Wet Wall Desuperheating	Select whether the process stream has wet wall desuperheating from the drop-down list: • Default (Yes) • Yes • No
Number of Points on Curve	Specify the number of points on the TASC stream heat load curve in this field. The minimum value is **6** and the maximum value is **12**.
Fit to Property Curve	Select whether the results fit the property curve from the drop-down list: • Default • A input / calc. • Use best fit

Method	Description
Subcooled Boiling	Select whether there is subcooled boiling from the drop-down list: • Default(ht.tr&pr.drop) • Allow in heat.tr&pr.drop • Allow in heat tran. only • Allow in press. drop only • Not allowed for
Post Dryout Heat Transfer	Select whether there is post dryout heat transfer from the drop-down list: • Default (allow) • Allow for • Assume Boiling
Pressure Drop Calculations	Select the type of pressure drop calculations from the drop-down list: • Default (fric+acc) • Frict+Acc+Gravitation • Friction+Accel
HTFS Colburn-Hougen Method	Select whether to apply HTFS Colburn-Hougen method from the drop-down list: • Default (no) • Yes • No
Downflow Condensate Cooling	Select the type of downflow condensate cooling from the drop-down list: • Default (standard) • Falling Film • Standard Method

Process Constraints Variables

If you select the Process Constraints radio button from the Methods and Constraints group, the Methods page appears as shown in the figure below:

Figure 4.95



The table below contains a list of the constraints available in the operation:

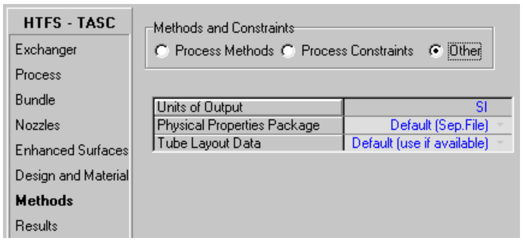
Refer to the **TASC Thermal Reference** guide for information about the selections available.

Constraints	Description
Revise for Heat Balance	Select the type of revise for heat balance from the drop-down list: • Default (h.load) • Heat Load • Outlet Temp. • Inlet Temp. • Flowrate
Liquid Heat Transfer Coefficient	Amount of liquid heat transfer coefficient.
Two Phase Heat Transfer Coefficient	Amount of two phase heat transfer coefficient.
Vapour Heat Transfer Coefficient	Amount of vapour heat transfer coefficient.
Liquid Heat Transfer Coefficient Multiplier	The liquid heat transfer coefficient multiplier.
Two Phase Heat Transfer Coefficient Multiplier	The two phase heat transfer coefficient multiplier.
Vapour Heat Transfer Coefficient Multiplier	The vapour heat transfer coefficient multiplier.
Pressure Drop Multiplier	The pressure drop multiplier.

Other Variables

If you select the Other radio button from the Methods and Constraints group, the Methods page appears as shown in the figure below.

Figure 4.96



The table below contains a list of variables available in the operation.

Refer to the **TASC Thermal Reference** guide for information about the selections available.

Variables	Description
Units of Output	Select the type of unit for the output from the drop-down list: • Default (as Input) • SI • British/US • Metric • unused option
Physical Property Package	Select the type of physical property package from the drop-down list: • Default (Sep.File) • In Lineprinter O/p • Separate File • No Output
Tube Layout Data	Select the type of tube layout data from the drop-down list: • Default (use if available) • Use if available • Revise from input • Ignore layout data

Results Page

The Heat Exchanger results appear on this page. The results are created in a text format that can be exported to HTFS-TASC.

4.5 LNG

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.

For the overall exchanger, you can specify various parameters, including heat leak/heat loss, UA or temperature approaches. Two solution approaches 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 that attempts to determine the solution that satisfies not only the energy balance, but also any constraints, such as temperature approach or UA.

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

The dynamic LNG exchanger model performs energy and material balances for a rating plate-fin type heat exchanger model. The dynamic LNG is characterized as having a high area density, typically allowing heat exchange even when low temperature gradients and heat transfer coefficients exist between layers in the LNG operation.

Some of the major features in the dynamic LNG operation include:

- A pressure-flow specification option which realistically models flow through the LNG operation according to the pressure network of the plant. Possible flow reversal situations can therefore be modeled.
- A dynamic model, which accounts for energy holdup in the metal walls and material stream layers. Heat transfer between layers depends on the arrangement of streams, metal properties, and fin and bypass efficiencies.
- Versatile connections between layers in a single or multiple zone LNG operation. It is possible to model cross and counter flow, and multipass flow configurations within the LNG operation.

- A heat loss model, which accounts for the convective and conductive heat transfer that occurs across the wall of the LNG operation.

4.5.1 Theory

Heat Transfer

The LNG calculations are based on energy balances for the hot and cold fluids. The following general relation applies any layer in the LNG unit operation.

$$M(H_{in} - H_{out}) + Q_{internal} + Q_{external} = \rho \frac{d(VH_{out})}{dt} \tag{4.37}$$

where:

M = *fluid flow rate in the layer*

ρ = *density*

H = *enthalpy*

Q_{internal} = *heat gained from the surrounding layers*

Q_{external} = *heat gained from the external surroundings*

V = *volume shell or tube holdup*

Pressure Drop

The pressure drop across any layer in the LNG unit operation can be determined in one of two ways:

- Specify the pressure drop.
- Define a pressure flow relation for each layer by specifying a k-value.

If the pressure flow option is chosen for pressure drop determination in the LNG, a k value is used to relate the frictional pressure loss and flow through the exchanger.

This relation is similar to the general valve equation:

$$f = \sqrt{density} \times k\sqrt{P_1 - P_2} \tag{4.38}$$

This general flow equation uses the pressure drop across the heat exchanger without any static head contributions. The quantity, P_1 - P_2, is defined as the frictional pressure loss which is used to “size” the LNG with a k-value.

Convective (U) & Overall (UA) Heat Transfer Coefficients

It is important to understand the differences between steady state and dynamics LNG models. The Steady State model is based on heat balances, and a number of specifications related to temperatures and enthalpy. In this model, the UA values are calculated based on heat curves. Whereas, the dynamic LNG model is a rating model, which means the outlet streams are determined by the physical layout of the exchanger.

Several of the pages in the LNG property view indicate whether the information applies to steady state or dynamics.

In steady state the order of the streams given to the LNG is not important but in the dynamics rating model the ordering of streams inside layers in each zone is an important consideration. The U value on the dynamics page of LNG refers to the convective heat transfer coefficient for that stream in contact with the metal layer.

For convenience, you can also specify a UA value in Dynamic mode for each layer, and it is important to note that this value is not an overall UA value as it is in steady state but accounts merely for the convective heat transfer of the particular stream in question with its immediate surroundings. These UA values are thus not calculated in the same way as in Steady State mode.

In Dynamic mode the U and UA value refers to the convective heat transfer (only) contribution between a stream and the metal that immediately surrounds it. The overall duty of each stream, in dynamic mode, is influenced by the presence of metal fins, fin efficiencies, direct heat flow between metal layers, and other factors, as it would be in a real plate-fin exchanger.

If you specify the convective UA values in Dynamic mode, than the size and metal holdup of the LNG are still considered.

Ideally in Dynamic mode the convective heat transfer coefficient, U, for each stream is specified. An initial value can be estimated from correlations commonly available in the literature or from the steady state UA values. The values specified can be manipulated by a spread sheet if desired. If the shut down and start up of the LNG is to be modeled, then the **U flow scaled** calculator should be selected on the Heat Transfer page, of the Rating tab, as it correctly scales the U values based on the flow.

If the streams in the rating model are properly laid to optimize heat transfer (in other words, arranged in the fashion hot-cold-hot-cold and not hot-hot-cold-cold on the Model page of the Dynamics tab), and the metal resistance is not significant and significant phase change is not taking place, then the UA values reported by steady state approximates the convective UA values that can be specified in Dynamic mode for the same results.

Dynamic Specifications

The following table lists the minimum dynamic specifications required for the LNG unit operation to solve:

Specification	Description
Zone Sizing	The dimensions of each zone in the LNG operation must be specified. All information in the Sizing page of the Rating tab must be completed. You can modify the number of zones in the Model page of the Dynamics tab.
Layer Rating	The individual layer rating parameters for each zone must be specified. All information on the Layers page of the Rating tab must be completed.
Heat Transfer	Specify an Overall Heat Transfer Coefficient, U, or Overall UA. These specifications can be made on the Heat Transfer page of the Rating tab.
Pressure Drop	Either specify an Overall Delta P or an Overall K-value for the LNG. Specify the Pressure Drop calculation method on the Specs page of the Dynamics tab.
Layer Connections	Every layer in each zone must be specified with one feed and one product. Complete the Connections group for each zone on the Model page of the Dynamics tab.

4.5.2 LNG Property View

There are two ways to add a LNG Exchanger to your simulation:

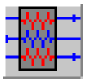
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12** .
2. Click the **Heat Transfer Equipment** radio button.
3. From the list of available unit operations, select LNG.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

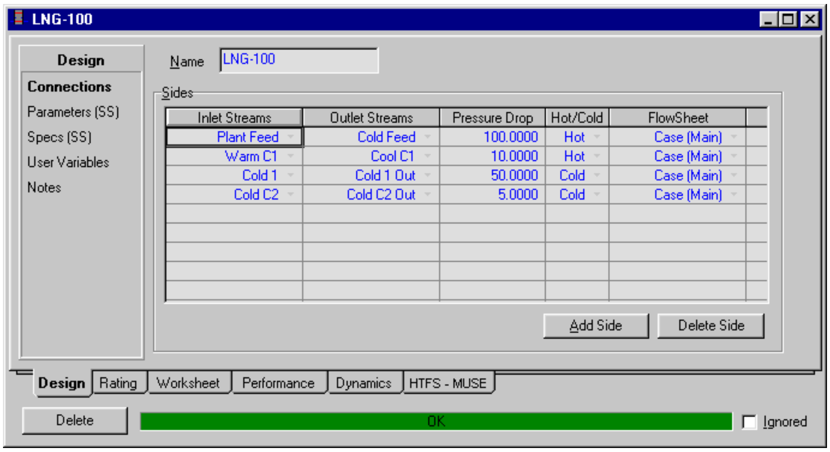
   You can also open the Object Palette by pressing **F4**.
2. Double-click the **LNG** icon.



LNG icon

The LNG property view appears.

Figure 4.97



To ignore the LNG during calculations, select the **Ignored** checkbox. HYSYS completely disregards the operation (and cannot calculate the outlet stream) until you restore it to an active state by clearing the checkbox.

4.5.3 Design Tab

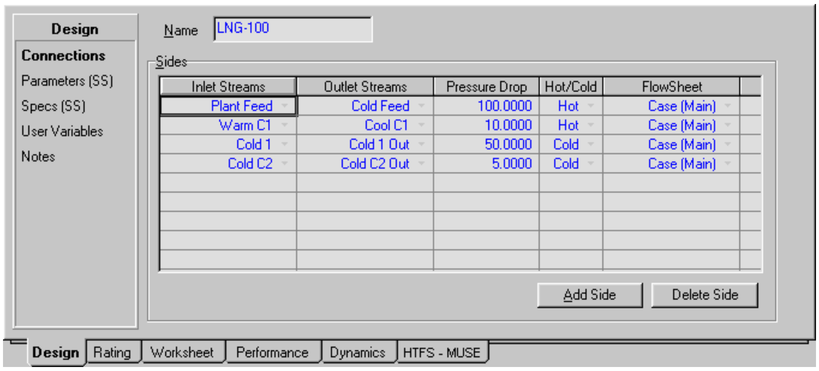
The Design tab contains the following pages:

- Connections
- Parameters
- Specs
- User Variables
- Notes

Connections Page

The Connections page is shown in the figure below.

Figure 4.98



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 is used as an estimate for calculations and is also used for drawing the PFD. If a designated hot pass is actually cold (or vice versa), the operation still solves properly. The actual Hot/Cold designation (as determined by the LNG) can be found on the Side Results page.

Any number of Sides can be added simply by clicking the Add Side button. To remove a side, select the side to be deleted and click the Delete Side button.

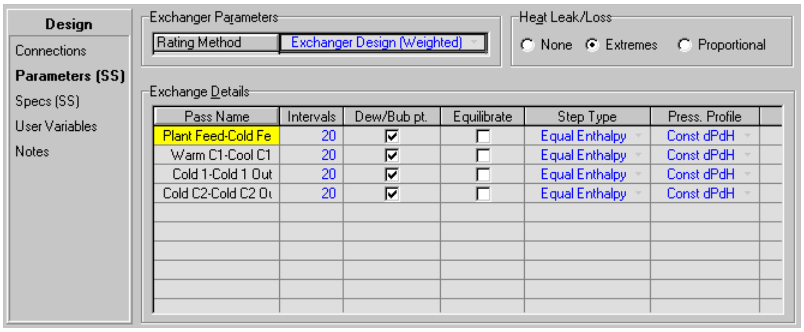
- The main flowsheet is the default shown in the flowsheet column.

The LNG status appears on the bottom of the property view, regardless of which page is currently shown. It displays an appropriate message such as Under Specified, Not Converged, or OK.

Parameters Page

On the Parameters page, you have access to the exchanger parameters, heat leak/loss options, the exchanger details, and the solving behaviour.

Figure 4.99



Exchanger Parameters Group

Parameters	Description
Rating Method	For the Weighted method, the heating curves are broken into intervals, which then exchange energy individually. An LMTD and UA are calculated for each interval in the heat curve and summed to calculate the overall exchanger UA.
Shell Passes	You have the option of having HYSYS perform the calculations for Counter Current (ideal with Ft = 1.0) operation or for a specified number of shell passes. You can specify the number of shell passes to be any integer between 1 and 7.

In Steady State mode, you can select either an End Point or Weighted Rating Method.

If there are more than two LNG sides, then only the Weighted rating method can be used.

Heat Leak/Loss Group

By default, the None radio button is selected. The other two radio buttons incorporate heat loss/heat leak:

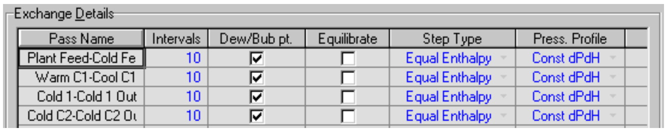
Radio Button	Description
Extremes	The heat loss and heat leak are considered to occur only at the end points (inlets and outlets) and are applied to the Hot and Cold Equilibrium streams.
Proportional	The heat loss and heat leak are applied over each interval.

Heat Leak/Loss group is available only when the Rating Method is Weighted.

Exchange Details Group

The LNG Exchange Details appear as follows:

Figure 4.100

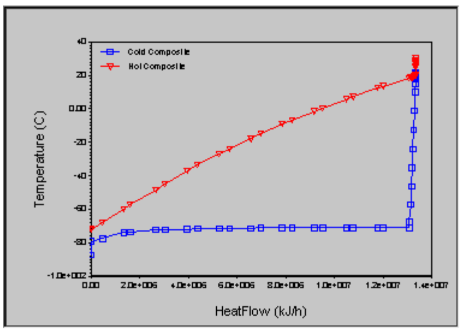


For each side, the following parameters can be specified:

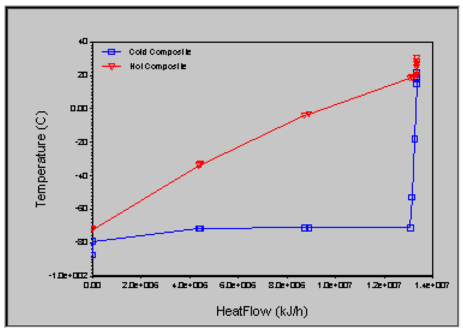
Parameter	Description
Intervals	The number of intervals, applicable only to the Weighted Rating Method, can be specified. For non-linear temperature profiles, more intervals are necessary.
Dew/Bubble Point	Select this checkbox to add a point to the Heat curve for a phase change. **Figure 4.101** illustrates the effect of the number of intervals and inclusion of the dew and bubble points on the temperature / heat flow curves. Temperature is on the y-axis, and heat flow is on the x-axis
Equilibrate	All sides that are checked comes to thermal equilibrium before entering into the UA and LMTD calculations. If only one hot stream or cold stream is checked, then that stream is by definition in equilibrium with itself and the results are not affected. If two or more hot or cold streams are checked, then the effective driving force is reduced. All unchecked streams enter the composite curve at their respective temperatures.

Parameter	Description
Step Type	There are three choices, which are described below. • **Equal Enthalpy**. All intervals have an equal enthalpy change. • **Equal Temperature**. All intervals have an equal temperature change. • **Auto Interval**. HYSYS determines where points should be added to the heat curve. This is designed to minimize the error, using the least amount of intervals.
Pressure Profile	The Pressure Profile is updated in the outer iteration loop, using one of the following methods described below. • **Constant dPdH**. Maintains constant dPdH during update. • **Constant dPdUA**. Maintains constant dPdUA during update. • **Constant dPdA**. Maintains constant dPdA during update. This is not currently applicable to the LNG Exchanger in steady state, as the area is not predicted. • **Inlet Pressure**. The pressure is constant and equal to the inlet pressure. • **Outlet Pressure**. The pressure is constant and equal to the pressure.

Figure 4.101



10 Intervals; Dew Bubble Points included

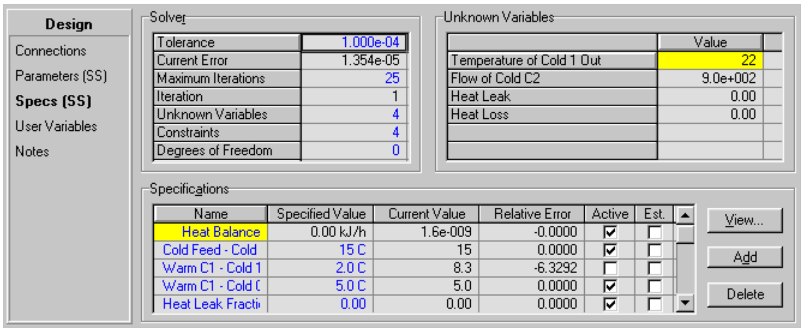


3 Intervals; Dew Bubble Points not included

Specs Page

On the Specs page, there are three groups which organize the various specification and solver information.

Figure 4.102



Solver Group

The Solver group includes the solving parameters used for LNG's:

Solver Parameter	Specification Description
Tolerance	You can set the calculation error tolerance.
Current Error	When the current error is less than the calculation tolerance, the solution is considered to have converged.
Maximum Iterations	You can specify the maximum number of iteration before HYSYS stops the calculations.
Iteration	The current iteration of the outer loop appears. In the outer loop, the heat curve is updated and the property package calculations are performed. Non-rigorous property calculations are performed in the inner loop. Any constraints are also considered in the inner loop.
Unknown Variables	Displays the number of unknown variables in the LNG.

Solver Parameter	Specification Description
Constraints	Displays the number specifications you have placed on the LNG.
Degrees of Freedom	Displays the number of Degrees of Freedom on the LNG. To help reach the desired solution, unknown parameters (flows, temperatures) can be manipulated in the attached streams. Each parameter specification reduces the Degrees of Freedom by one. The number of Constraints (specs) must equal the number of Unknown Variables. When this is the case, the Degrees of Freedom is equal to zero, and a solution is calculated.

Unknown Variables Group

HYSYS lists all unknown LNG variables according to your specifications. Once the unit has solved, the values of these variables appear.

Specifications Group

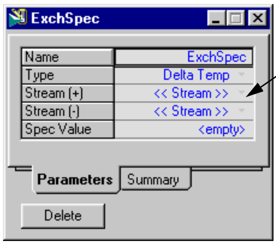
Notice the Heat Balance (specified at **0 kJ/h**) is considered to be a constraint. This is a Duty Error spec; if you turn it off, the heat equation cannot balance. Without the Heat Balance spec, you can, for example, completely specify all four heat exchanger streams, and have HYSYS calculate the Heat Balance error, which would be displayed in the Current Value column of the Specifications group.

The Heat Balance specification is a default LNG specification that must be active for the heat equation to balance.

You can view or delete selected specifications by using the buttons that align the right of the group. A specification property view appears automatically each time a new spec is created via the Add button.

In the figure below is a typical property view of a specification, which is accessed via the **View** or **Add** button.

Figure 4.103



As an example, defining the Delta Temp Spec requires two stream names, and a value for the specification.

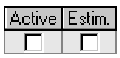
Each specification property view has two tabs:

- Parameters
- Summary

The Summary page is used to define whether the specification is Active or an Estimate. The Spec Value is also shown on this page.

Information specified on the Summary page of the specification property view also appears in the Specifications group.

All specifications are one of the following three types:

Specification Type	Action
Active 	An active specification is one which the convergence algorithm is trying to meet. Notice an active specification always serves as an initial estimate (when the **Active** checkbox is selected, HYSYS automatically selects the **Estimate** checkbox). An active specification exhausts one degree of freedom. An Active specification is one which the convergence algorithm is trying to meet. Both checkboxes are selected for this specification.
Estimate 	An estimate is considered an Inactive specification because the convergence algorithm is not trying to satisfy it. To use a specification as an estimate only, clear the **Active** checkbox. The value then serves only as an initial estimate for the convergence algorithm. An estimate does not use an available degree of freedom. An Estimate is used as an “initial guess” for the convergence algorithm, and is considered to be an Inactive specification.
Completely Inactive 	To disregard the value of a specification entirely during convergence, clear both the **Active** and **Estimate** checkboxes. By ignoring rather than deleting a specification, it is available if you want to use it later or simply view its current value. A Completely Inactive specification is one which is ignored completely by the convergence algorithm, but can be made Active or an Estimate at a later time.

The specification list allows you to try different combinations of the above three specification types. For example, suppose you have a number of specifications, and you want to determine which ones should be active, which should be estimates and which ones should be ignored altogether. By manipulating the checkboxes among various specifications, you can test various combinations of the three types to see their effect on the results.

The available specification types are:

Specification	Description
Temperature	The temperature of any stream attached to the LNG. The hot or cold inlet equilibrium temperature can also be defined.
Delta Temp	The temperature difference at the inlet or outlet between any two streams attached to the LNG. The hot or cold inlet equilibrium temperatures can also be used.

Specification	Description
Minimum Approach	The minimum temperature difference between the specified pass and the opposite composite curve. For example, if you select a cold pass, this is the minimum temperature difference between that cold pass and the hot composite curve. • The Hot Inlet Equilibrium temperature is the temperature of the inlet hot stream minus the heat loss temperature drop. • The Cold Inlet Equilibrium temperature is the temperature of the inlet cold stream plus the heat leak temperature rise.
UA	The overall UA (product of overall heat transfer coefficient and heat transfer area).
LMTD	The overall log mean temperature difference. It is calculated in terms of the temperature approaches (terminal temperature differences) in the exchanger. See **Equation (4.39)**.
Duty	The overall duty, duty error, heat leak or heat loss. The duty error should normally be specified as **0** so that the heat balance is satisfied. The heat leak and heat loss are available as specifications only if Heat Loss/Leak is set to Extremes or Proportional on the Parameters page.
Duty Ratio	A duty ratio can be specified between any two of the following duties: overall, error, heat loss, heat leak or any pass duty.
Flow	The flowrate of any attached stream (molar, mass or liquid volume).
Flow Ratio	The ratio of any two inlet stream flowrates.

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

4.5.4 Rating Tab

While working exclusively in Steady State mode, you are not required to change any information on the pages accessible through this tab.

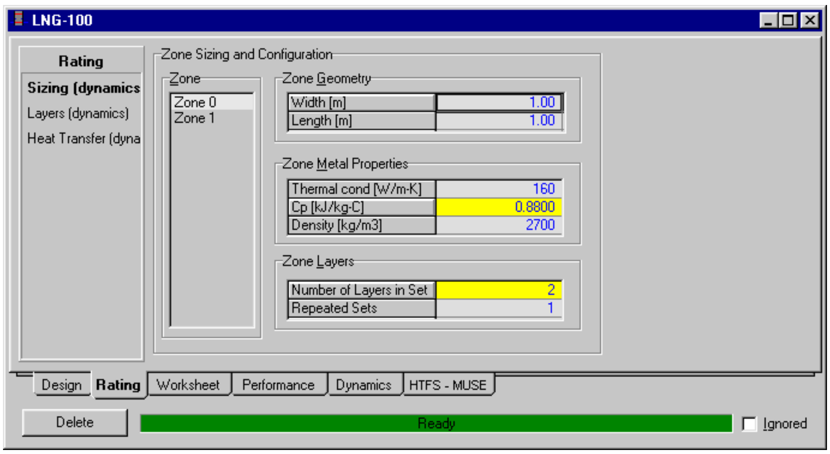
The Rating tab contains the following pages:

- Sizing (dynamics)
- Layers (dynamics)
- Heat Transfer (dynamics)

Sizing (dynamics) Page

On the Sizing (dynamics) page, you can specify the geometry of each zone in the LNG unit operation.

Figure 4.104

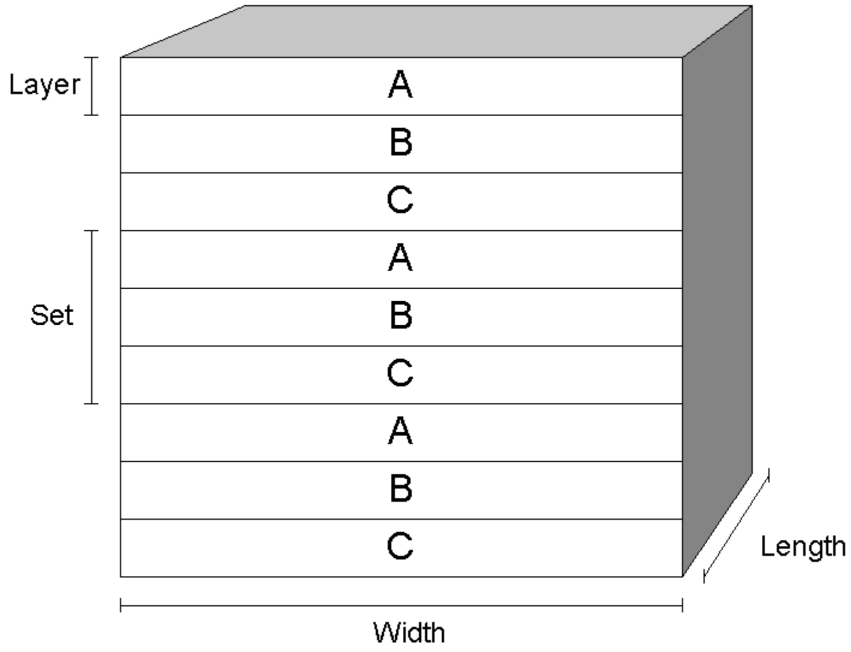


You can partition the exchanger into a number of zones along its length. Each zone features a stacking pattern with one feed and one product connected to each representative layer in the pattern.

In practice, a plate-fin heat exchanger may have a repeating pattern of layers in a single exchanger block. A set is defined as a single pattern of layers that are repeated over the height of an exchanger block. Each zone can be characterized with a multiple number of sets each with the same repeating pattern of layers.

The figure below displays an LNG exchanger block (zone) with 3 sets, each containing 3 layers:

Figure 4.105



The Zone Sizing and Configuration group contains information regarding the geometry, heat transfer properties, and configuration of each zone in the LNG unit operation. To edit a zone, select the individual zone in Zone group, and make the necessary changes to the other groups.

The Zone Geometry group displays the following information regarding the dimensions of each zone:

- Width
- Length

This length refers to the actual length of the exchanger, which is used for heat transfer. The remainder is taken up by the flow distributors. The flow of material travels in the direction of the length of the exchanger block. The fins within each layer are situated across the width of the exchanger block.

The Zone Metal Properties group contains information regarding the metal heat transfer properties:

- Thermal Conductivity
- Specific Heat Capacity, Cp
- Density

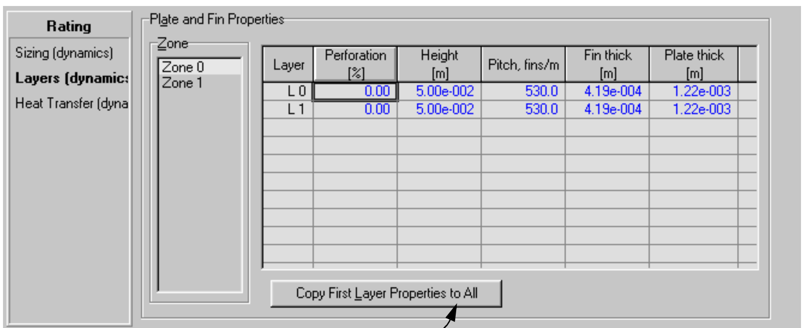
The Zone Layers group contains the following information regarding the configuration of layers in the zone:

- Number of Layers in a Set
- Repeated Sets

Layers (dynamic) Page

The Layers (dynamics) page contains information regarding the plate and fin geometry:

Figure 4.106



The Copy First Layer Properties to All button can be clicked if you want to specify all the layers in the zone with the same plate and fin properties.

Each of the following plate and fin properties should be specified for every layer in each zone if the LNG operation is to solve:

Plate and Fin Property	Description
Fin Perforation	The perforation percentage represents the area of perforation relative to the total fin area. Increasing the Fin Perforation decreases the heat transfer area.
Height	The height of the individual layers. This affects the volume of each layer holdup.
Pitch	The pitch is defined as the fin density of each layer. The pitch can be defined as the number of fins per unit width of layer.
Fin thickness	The thickness of the fin in the layer.
Plate thickness	The thickness of the plate.

Heat Transfer (dynamics) Page

The Heat Transfer (dynamics) page displays the heat transfer coefficients associated with the individual layers of the LNG unit operation. You can select internal or external heat transfer by selecting the appropriate Heat Transfer radio button.

HYSYS accounts for the heating and cooling of the metal fins and plates in the LNG unit operation. The calculation of heat accumulation in the metal is based on the conductive heat transfer properties, fin efficiencies, and various other correction factors. An initial metal temperature can be specified for each zone in the **Initial Metal Temperature** field.

Since a repeating stacking pattern is used, the top most layer of a set is assumed to exchange heat with the bottom layer of the set above.

You can also select the Brazed Aluminum Plate-Fin heat transfer calculation standards by selecting the **Calculate fin area using the standards of the Brazed Aluminium Plate-Fin HX Manufacturer’s Association** checkbox.

Select the **Auto Prevent Temp. Cross** checkbox to enter two parameters for split steps, and prevent the temperature from crossing along the heat transfer passes.

Select the **Automatically Update k's** checkbox to automatically update the k's based on current relationships between P-F flow rates and pressure drops for all the heat transfer layers, making the LNG steam flow rates more stable.

LNG Temperature Crossing Project

The LNG Temperature Crossing Project redistributes the zone length fractions among the total flow pass length and multiple zones to prevent the temperature from crossing along the heat transfer passes.

It uses a cascade of lumping heat zones to incorporate the distributed systems, and requires at least 10 zones to automatically remove the big temperature wiggle profiles within the flow passes. Under certain conditions, such as zone number and the changes in temperature and flow rates, the original function of Auto Prevent Temp Cross could smooth the small temperature waves. But it also made the dynamic processes unstable.

To minimize temperature and flow instability in the LNG dynamic processes:

1. Specify 10 or more heat zones to remove the wiggle temperature profiles.
2. Select the **Automatically Update k's** checkbox to make the LNG flow rates more stable if your LNG flow rates are not too small.
3. Select the **Auto Prevent Temp Cross** checkbox to prevent temperature cross and lessen small temperature waves .
4. Use the following parameters for the Auto Prevent Temperature Crossing:
    - Reach small split steps
        - A smaller value (0.001-1000) helps to prevent small temperature crossing.
    - Reach even split steps
        - A small value (0.1-1000) leads to a quick speed.

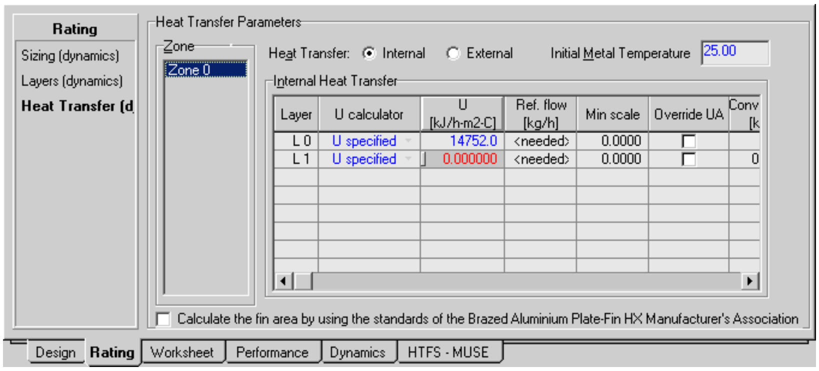
Figure 4.107

!!! Need to capture the screen shot again!!!

Internal Heat Transfer

If you select the Internal radio button, the internal heat transfer coefficient associated with each layer appears as shown in the figure below.

Figure 4.108



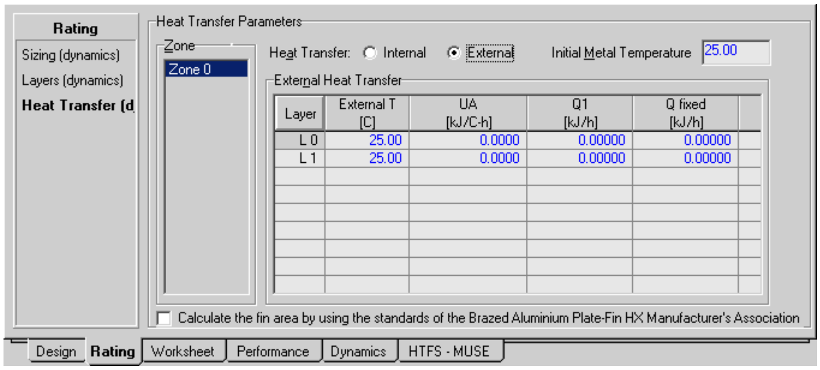
Currently, the internal heat transfer coefficient, U, or the overall UA must be specified for the LNG unit operation. HYSYS cannot calculate the heat transfer coefficient from the geometry/configuration of the plates and fins. The Internal Heat Transfer group contains the following parameters:

Parameter	Description
U Calculator	The heat transfer calculator currently available in HYSYS are U specified and U flow scaled. If U specified is selected, you must specify the internal heat transfer coefficient, U. Alternatively, you can select U flow scaled calculator and a reference flow rate is used to calculate U.
U	The internal heat transfer coefficient is specified in this cell.
Ref. Flow	The Reference Flow is used to calculate U when the U Flow Scaled calculator is selected.
Min Scale	The minimum scale factor is applied to U by the U Flow Scaled calculator when the flow changes.
Override UA	The overall UA can be specified if the **Override UA** checkbox is selected. The specified UA value is used without the consideration or back calculation of the internal heat transfer coefficient, U.
Convective UA	The overall UA is specified in this cell.

External Heat Transfer

If you select the External radio button, the overall UA associated with heat loss to the atmosphere appears.

Figure 4.109



Like the internal heat transfer coefficients, the external overall UA must be specified.The External Heat Transfer group contains the following parameters:

Parameter	Description
External T	The ambient temperature surrounding the plate-fin heat exchanger. This parameter may be specified or can remain at its default value.
UA	The overall UA is specified in this field. The heat gained from the ambient conditions is calculated using the overall UA.
Q1	Q1 is calculated from the overall UA and the ambient temperature. If heat is gained in the holdup, Q1 is positive; if heat is lost, Q1 is negative.
Qfixed	A fixed heat value can be added to each layer in the LNG unit operation. Since Qfixed does not vary, a constant heat source or sink is implied (for example, electrical tracing). If heat is gained in the holdup, Qfixed is positive; if heat is lost, Qfixed is negative.

4.5.5 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the LNG unit operation.

The PF Specs page is relevant to dynamics cases only.

4.5.6 Performance Tab

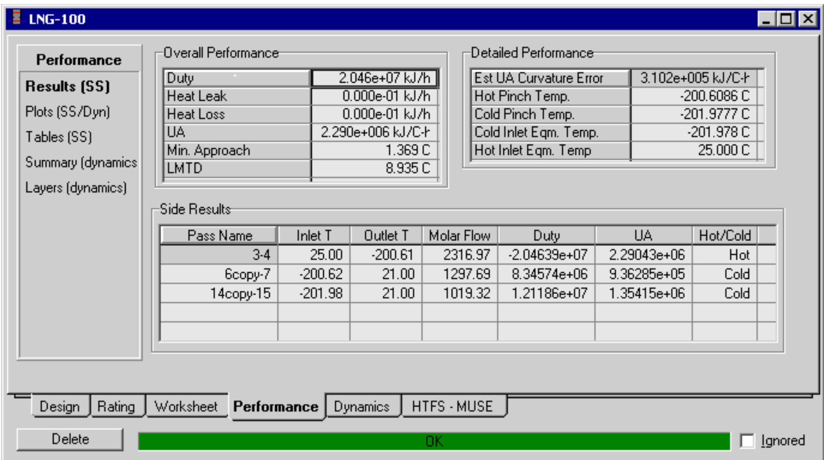
The Performance tab contains detail performance results of the LNG exchanger. The calculated results are displayed in the following pages:

- Results (SS). Contains information relevant only to Steady State mode.
- Plots (SS/Dyn). Contains information relevant to both Steady State and Dynamics mode.
- Tables (SS). Contains information relevant only to Steady State mode.
- Summary (dynamics). Contains information relevant only to Dynamics mode.
- Layers (dynamics). Contains information relevant only to Dynamics mode.

Results Page

The Results page displays the calculated values generated by HYSYS. These values are split into three groups for your convenience.

Figure 4.110



Overall Performance Group

Parameter	Description
Duty	Combined heat flow from the hot streams to the cold streams minus the heat loss. Conversely, this is the heat flow to the cold streams minus the heat leak.
Heat Leak	Loss of cold side duty to leakage.
Heat Loss	Loss of hot side duty to leakage.
UA	Product of the Overall Heat Transfer Coefficient and the Total Area available for heat transfer. The LNG Exchanger duty is proportional to the overall log mean temperature difference, where UA is the proportionality factor. That is, the UA is equal to the overall duty divided by the LMTD.
Minimum Approach	The minimum temperature difference between the hot and cold composite curves.
LMTD	The LMTD is calculated in terms of the temperature approaches (terminal temperature differences) in the exchanger, using **Equation (4.39)**.

The equation used to calculate LMTD is:

$$\Delta T_{LM} = \frac{\Delta T_1 - \Delta T_2}{\ln(\Delta T_1 / \Delta T_2)} \tag{4.39}$$

where:

$$\Delta T_1 = T_{hot,\ out} - T_{cold,\ in}$$

$$\Delta T_2 = T_{hot,\ in} - T_{cold,\ out}$$

Detailed Performance Group

Parameter	Description
Estimated UA Curvature Error	The LMTD is ordinarily calculated using constant heat capacity. An LMTD can also be calculated using linear heat capacity. In either case, a different UA is predicted. The UA Curvature Error reflects the difference between these UAs.
Hot Pinch Temperature	The hot stream temperature at the minimum approach between composite curves.
Cold Pinch Temperature	The cold stream temperature at the minimum approach between composite curves.
Cold Inlet Equilibrium Temperature	The Equilibrium Temperature for the cold streams. When streams are not equilibrated (see the Parameters page), the Equilibrium temperature is the coldest temperature of all cold inlet streams.
Hot Inlet Equilibrium Temperature	The Equilibrium Temperature for the hot streams. When streams are not equilibrated (see the Parameters Page), the Equilibrium temperature is the hottest temperature of all hot inlet streams.

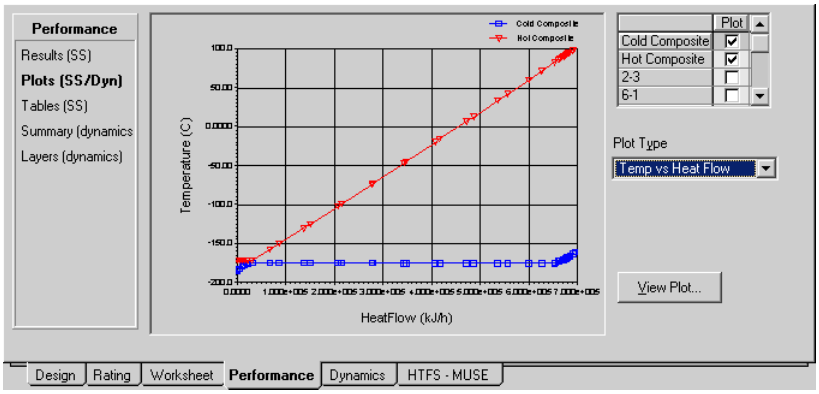
Side Results Group

The Side Results group displays information on each Pass. For each side, the inlet and outlet temperatures, molar flow, duty, UA, and the hot/cold designation appear.

Plots Page

On the Plots page, you can plot composite curves or individual pass curves for the LNG. The options available on this page varies, depending on the type of mode (Steady State or Dynamics) your simulation case is in.

Figure 4.111



Refer to **Section 1.3.1 - Graph Control Property View** for more information.

You can modify the appearance of the plot via the Graph Control property view.

Use the checkboxes under the **Plot** column to select which curve(s) you want to appear in the plot.

- In Steady State mode, all the checkboxes under the **Plots** column are active.
- In Dynamics mode, the **Cold Composite** and **Hot Composite** checkboxes are unavailable.

The data displayed in the plot varies depending on the simulation mode:

- In Steady State mode, the information in the plot is controlled by the selection in the **Plot Type** drop-down list.

The Plot Type drop-down list is only available at Steady State mode.

The **Plot Type** drop-down list enables you to select any combination of the following data for the x and y axes: Temperature, UA, Delta T, Enthalpy, Pressure, and Heat Flow.

- In the Dynamics mode, the plot only displays the Temperature vs. Zone data.

Use the **View Plot** button to open the plot area in a separate property view.

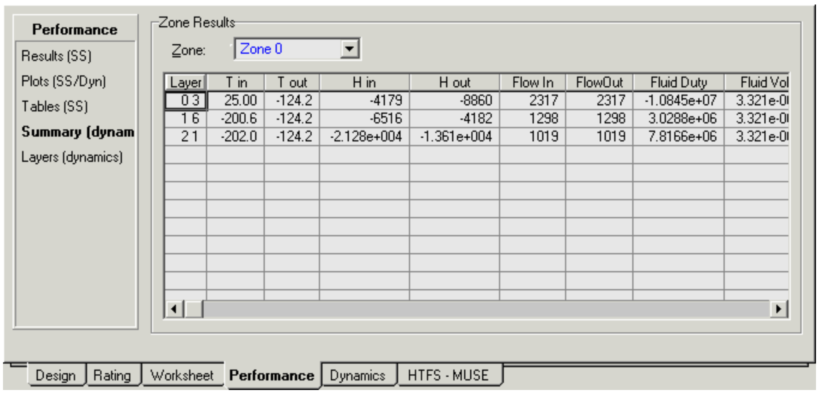
Tables Page

On the Table page, you can examine the interval Temperature, Pressure, Heat Flow, Enthalpy, UA, Vapour Fraction, and Delta T for each side of the Exchanger in a tabular format. Choose the side, Cold Composite or Hot Composite, by making a selection from the **Side** drop-down list located above the table.

Summary Page

The Summary page displays the results of the dynamic LNG unit operation calculations.

Figure 4.112



On this page, the following zone properties appear for each layer:

- Layer
- Inlet Temperature

- Exit Temperature
- Inlet Enthalpy
- Exit Enthalpy
- Inlet Flow rate
- Outlet Flow rate
- Fluid Duty
- Fluid Volume
- Surface Area
- Metal Mass

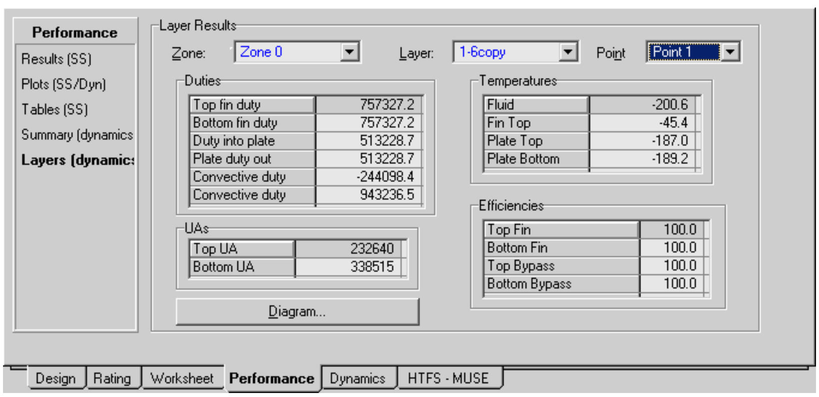
The Fluid Duty is defined as the energy specified to the holdup. If the fluid duty is positive, the layer gains energy from its surroundings; if the fluid duty is negative, the layer loses energy to its surroundings.

If the Combine Layers checkbox is selected in the Model page of the Dynamics tab, some parameters in the Summary page of the Performance tab include contributions from multiple layers.

Layers Page

The Layers page displays information regarding local heat transfer and fluid properties at endpoint locations in each layer of each zone.

Figure 4.113



The information displayed on this page is not central to the performance of the LNG operation.

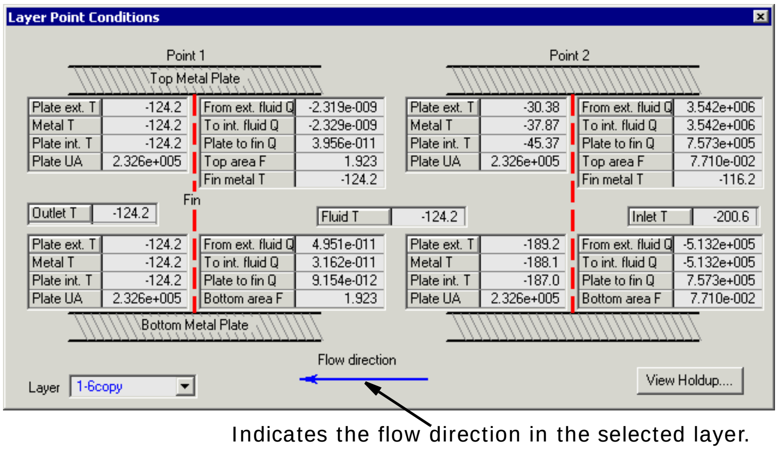
Use the **Zone**, **Layer**, and **Point** drop-down list to select the zone, layer, and endpoint location you want to see.

Click the **Diagram** button to access the Layer Point Conditions property view.

Layer Point Conditions Property View

The Layer Point Conditions property view displays the detailed temperatures and overall heat transfer values for both endpoints of the selected layer.

Figure 4.114



You can select a different layer using the **Layer** drop-down list.

Refer to **Section 1.3.4 - HoldUp Property View** for more information.

Click the **View Holdup** button to access the Holdup property view.

4.5.7 Dynamics Tab

The Dynamics tab contains the following pages:

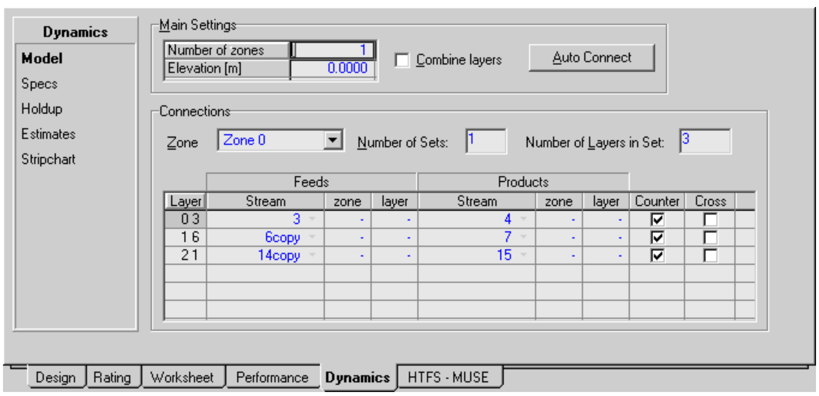
- Model
- Specs
- Holdup
- Estimates
- Stripchart

If you are working exclusively in steady state mode, you are not required to change any information on the pages accessible through this tab.

Model Page

On the Model page, you can specify how each layer in a multi-zone LNG unit operation is connected.

Figure 4.115



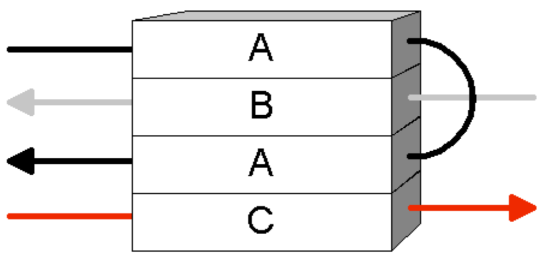
Main Settings

The Main Settings group displays the following LNG model parameters:

Parameter	Description
Number of Zones	The number of zones in a LNG unit operation can be specified in this field.
Elevation	You can specify the elevation of the LNG in this field. The elevation is significant in the calculation of static head in and around the LNG unit operation.
Combine Layers Checkbox	With the **Combine Layers** checkbox selected, individual layers (holdups) carrying the same stream in a single zone is calculated using a single holdup. The Combine Layers option increases the speed of the dynamic solver, and usually yields results that are similar to a case not using the option.

The Connections group displays the feed and product streams of each layer for every zone in the LNG unit operation. Every layer must have one feed stream and one product stream in order for the LNG operation to solve. A layer's feed or product stream can originate internally (from another layer) or externally (from a material stream in the simulation flowsheet). Thus, various different connections can be made allowing for the modeling of multi-pass streams in a single zone.

Figure 4.116



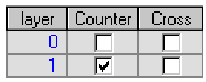
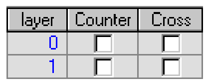
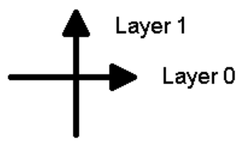
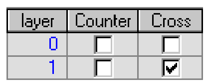
Connections Group

Every zone in the LNG unit operation is listed in the Zone drop-down list in the Connections group. All the layers in the selected zone in one set appear. For every layer's feed and product, you must specify one of the following:

- An external material stream.
- The zone and layer of an internal inlet or exit stream.

You can specify the relative direction of flow in each layer in the zone. Layers can flow counter (in the opposite direction) or across the direction of a reference stream. The reference stream is defined as a stream which does not have either the **Counter** or **Cross** checkbox selected in the Connections group.

The following table lists three possible flow configurations:

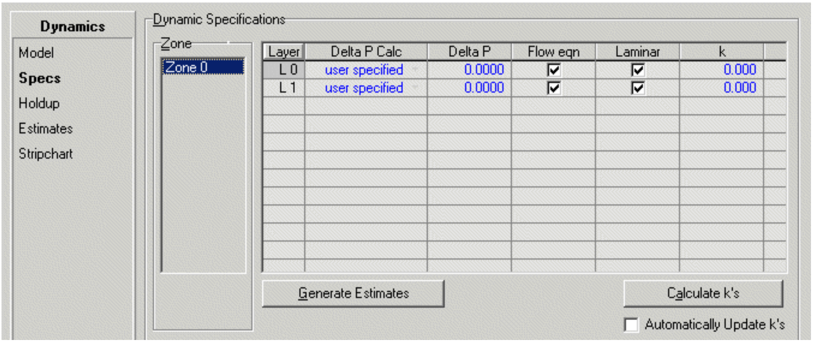
Description	Flow Direction	Flow Setting
Counter Current Flow		
Parallel Flow		
Cross Flow		

To implement counter current flow for two streams in a single exchanger block, ensure that the Counter checkbox is selected for only one of the streams. If the Counter checkbox is selected for both streams, the flow configuration is still parallel, and in the opposite direction.

Specs Page

The Specs page contains information regarding the calculation of pressure drop across the LNG unit operation.

Figure 4.117



The following parameters appear for every layer in each zone in the LNG unit operation in the Dynamic Specification groups.

Dynamic Specification	Description
Delta P Calculator	The Delta P Calculator allows you to either specify or calculate the pressure drop across the layer in the LNG operation. Specify the cell with one of following options: • **user specified**. You specify the pressure drop. • **not specified**. Pressure drop across the layer is calculated from a pressure flow relationship. You must specify a k-value, and select the **Flow Eqn** checkbox if you want to use this non specified Delta P calculator.
Delta P	The pressure drop across the layer of the LNG operation can be specified or calculated.
Flow eqn	Activate this option, if you want to have the Pressure Flow k value used in the calculation of pressure drop. If the **Flow Eqn** checkbox is selected, the Delta P calculator must also be set to not specified.

Dynamic Specification	Description
Laminar	HYSYS is able to model laminar flow conditions in the layer. Select the **Laminar** checkbox if the flow through the layer is in the laminar flow regime.
Pressure Flow k Value	The k-value defines the relationship between the flow through layer and the pressure of the surrounding streams. You can either specify the k-value or have it calculated from the stream conditions surrounding the layer. You can “size” each layer in the zone with a k-value by clicking the Calculate k’s button. Ensure that there is a non zero pressure drop across the LNG layer before the Calculate k button is clicked. Each zone layer can be specified with a flow and set pressure drop by clicking the Generate Estimates button. The LNG unit operation, like other dynamic unit operations, should use the k-value specification option as much as possible to simulate actual pressure flow relations in the plant.

When you click the Generate Estimates button, the initial pressure flow conditions for each layer are calculated. HYSYS generates estimates using the assumption that the flow of a particular stream entering the exchanger block (zone) is distributed equally among the layers. The generated estimates appear on the Estimates page of the Dynamics tab. It is necessary to complete the Estimates page in order for the LNG unit operation to solve.

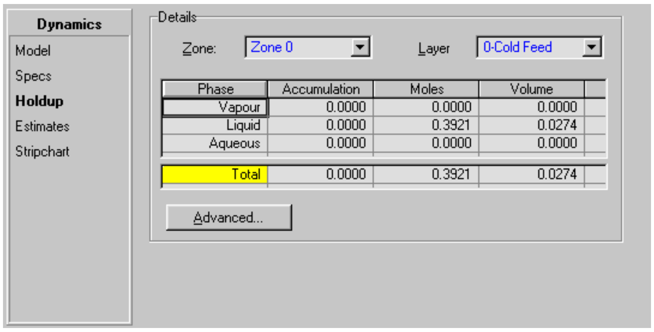
It is strongly recommended that you specify the same pressure drop calculator for layers that are connected together in the same exchanger block or across adjacent exchanger blocks. Complications arise in the pressure flow solver if a stream’s flow is set in one layer, and calculated in the neighbouring layer.

The Automatically Update k’s checkbox automatically updates the k’s based on current relationships between P-F flow rates and pressure drops for all the heat transfer layers, making the LNG steam flow rates more stable.

Holdup Page

The Holdup page contains information regarding each layer's holdup properties, composition, and amount.

Figure 4.118



Refer to **Section 1.3.3 - Holdup Page** for more information.

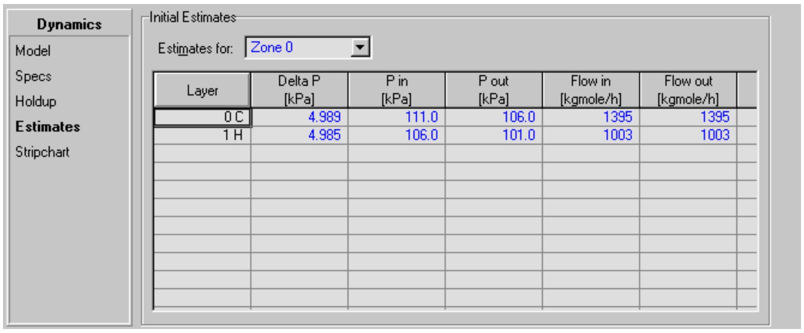
The Details group contains detailed holdup properties for every layer in each zone of the LNG. In order to view the advanced properties for individual holdups, you must first select the individual holdup.

To choose individual holdups you must specify the Zone and Layer in the corresponding drop-down lists.

Estimates Page

The Estimates page contains pressure flow information surrounding each layer in the LNG unit operation:

Figure 4.119



The following pressure flow information appears on the Estimates page:

- Delta P
- Inlet Pressure
- Outlet Pressure
- Inlet Flow
- Outlet Flow

It is necessary to complete the Estimates page in order for the LNG unit operation to completely solve. The simplest method of specifying the Estimates page with pressure flow values is having HYSYS estimate these values for you. This is achieved by clicking the Generate Estimates button on the Specs page of the Dynamics tab. HYSYS generates estimates using the assumption that the flow of a particular stream entering the exchanger block (zone) is distributed equally among the layers.

Stripchart Page

Refer to **Section 1.3.7 - Stripchart Page / Tab** for more information.

The Stripchart page allows you to select and create default strip charts containing various variable associated to the operation.

4.5.8 HTFS-MUSE Tab

The HTFS-MUSE tab integrates the HTFS' MUSE application into the HYSYS LNG unit Calculation. HYSYS can use the MUSE and MULE calculation Engines.

MUSE can perform a range of calculations on plate-fin heat exchangers, either simple two-stream exchangers, or complex ones with multiple streams. The basic calculation options are described in the table below:

Calculation Modes	Description
Simulation	Determines the heat load, pressure changes and outlet conditions for each stream in the exchanger, based on an exchanger you specify, and given stream inlet conditions.
Layer by Layer Simulation	For the simulation of a plate fin heat exchanger on a layer by layer basis. It must be specified with a layer pattern. It predicts temperature profiles through the layer pattern, which can be used to assess how good the layer pattern is.
Thermosyphon	Determines the performance of an exchanger, with a geometry you specify, with one stream operating as a thermosyphon. The exchanger can either be internal to the column or outside it and connected via pipe work. You can specify either the head of liquid driving the thermosyphon flow, or the thermosyphon stream flowrate, leaving the program to calculate the one you do not specify.
Design	Produces a "first shot" design of a heat exchanger to meet a heat load duty and pressure drop limits, which you specify for each stream. This should be a useful indication of what a specialist manufacturer would provide. A final design of a plate-fin exchanger must, however, come from a manufacturer, who can use proprietary finning and specialist design and manufacturing techniques.

These calculation types all relate to co- or counter-current exchangers.

The HTFS-MUSE tab contains two buttons:

- **Import**. Allows you to import values from MUSE into the pages of the tab.
- **Export**. Allows you to export the information provided within this tab to MUSE.

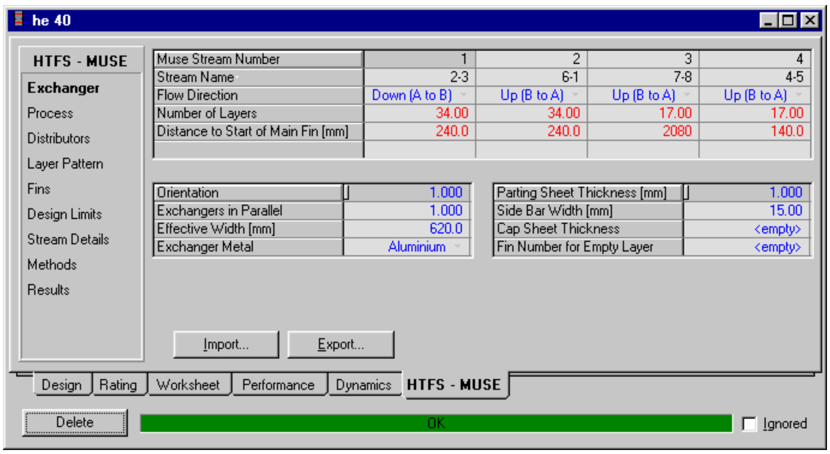
The HTFS-MUSE tab contains the following pages:

- Exchanger
- Process
- Distributors
- Layer Pattern
- Fins
- Design Limits
- Stream Details
- Methods
- Results

Exchanger Page

The Exchanger page allows you to specify parameters that define the geometric configuration of the exchanger, as well as the stream.

Figure 4.120



The group located on the top of the page is for specifying the stream geometry and consists of the following fields:

Field	Description
Flow Direction	There are two options that you can choose from to define the Flow direction of the stream. • flow away from end A, Up (B to A). • flow towards end A, Down (A to B). Normal design practice is for hot streams to flow away from end A (which is at the top of the exchanger), while cold streams flow towards end A.
Number of Layers	Allows you to enter the total number of layers a stream occupies in the exchanger. If there is more than one exchanger in parallel, enter the number for one exchanger only. This item can be omitted if a layer pattern is specified. If you specify both, however, they are cross-checked, which can be useful in detecting errors in a layer pattern input. When they are inconsistent, a warning is produced. If a stream is re-distributed, and occupies extra layers for part of its length, enter the basic number of layers only here, and specify the additional layers on the Distributors page.
Distance to Start of Main Fin	Allows you to enter the distance to the start of a stream's main finning from the fixed reference point. If omitted, the default is zero. If this distance is less than the distance to the start of the effective length, then there is a region of main fin where pressure drop, but no heat transfer is evaluated. If this distance is greater than that to the start of the effective length, then the stream has a draw-on or draw-off point part way along the exchanger.

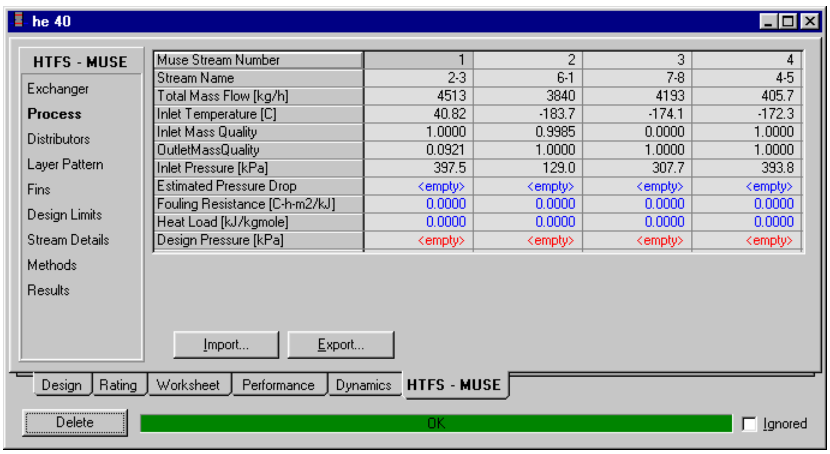
The remainder of the groups located on this page are for specifying the exchangers geometry and consists of the following fields:

Field	Description
Orientation	Plate Fin heat exchangers are normally vertical, with flow up or down. Enter 1.0 for vertical exchangers with the reference end, A, at the top. For horizontal or inclined exchangers, refer to the MUSE help file.
Exchangers in Parallel	More than one exchanger in parallel can be used when stream flowrates, or thermal duties are too large to be handled by a single exchanger. In all cases the exchangers are assumed to be identical, and no calculations are performed on pressure losses in connecting pipe work.
Effective Width	The effective flow width is the total width of the exchanger less the widths of the two side bars.

Field	Description
Exchanger Metal	Plate Fin exchangers for LNG and other cryogenic duties are made of aluminium. For other exchangers, you can select from: • aluminium • stainless steel • titanium
Parting Sheet Thickness	The thickness of the separating plates (parting sheets) between layers is used to determine the exchanger stack height, and also has an effect on the metal resistance to heat transfer.
Side Bar Width	Side bars form the sides and ends of each layer. This item does not usually affect the calculated results, with the exception of longitudinal conduction calculations.
Cap Sheet Thickness	The stack height is the sum of the fin heights and parting sheet thicknesses for every layer in the exchanger, plus the thickness of the two side plates (cap sheets).
Fin Number for Empty Layer	If you specify a layer pattern with some layers containing no streams enter the fin number to identify the fins used in such layers.

Process Page

Figure 4.121



The Process page allows you to specify the following process information for the streams attached to your exchanger:

- Estimated Pressure Drop
- Fouling Resistance

- Heat Load
- Design Pressure

Distributors Page

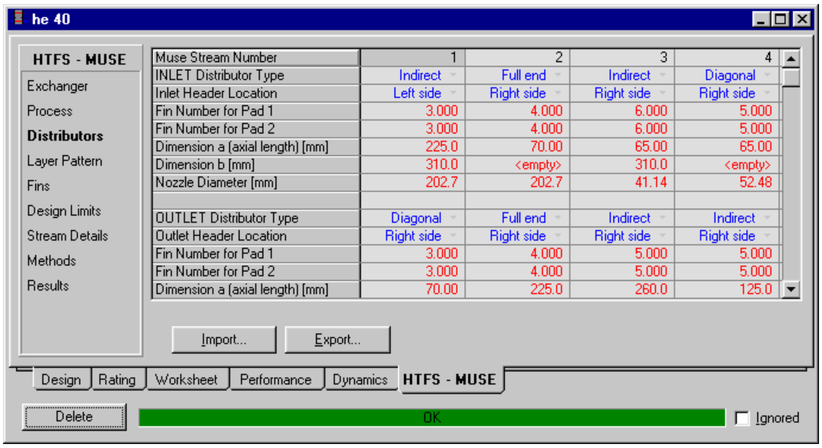
Distributors are special regions of finning, usually laid at an angle, that directs the flow between a header (inlet or outlet) and the main heat transfer finning. A low frequency perforated fin is usually used - for example 6fpi, 25% perforated. Distributor data is optional for each stream. If omitted the distributor pressure drop for that stream is ignored.

Heat transfer in inlet and outlet distributors is not considered, but if all streams have distributor data specified, an estimate is made of the heat transfer margin associated with each distributor. When distributor pressure drops are calculated, an estimate is made of the risk of maldistribution across the width of each layer.

When Redistribution is used, you should specify the Redistributor, and corresponding re-inlet distributor. Redistributors associated with partial draw-off of streams can also be specified.

Each distributor type consists of a set of inputs as shown in the figure below.

Figure 4.122



- Inlet/Outlet Distributor Type

Field	Description
Type	Allows you to specify the redistributor type, and the side of the exchanger on which the associated header is located. You have seven options: • Full End • End-Side • Central • Diagonal • Mitred • Indirect • Hardway
Header Location	Allows you to specify the side of the exchanger that the header is located. You have four options: • Right side • Left side • Central • Twin
Fin Number for Pad 1 and 2	Numbers to identify the fins used in the inlet/outlet distributor pads. Distributors typically use 6fpi 255 perforated finning. The same fin is usually used in both pads, so only pad 1 need normally be specified. Pad 1 is adjacent to the header.
Dimension a (axial length)	Dimension a for the inlet/outlet distributor. Dimension a is the length along the exchanger occupied by the distributor.

Field	Description
Dimension b	Dimension b for the inlet/outlet distributor. This is the header diameter for End Side, Central, Indirect and Hardway distributors, and the Pad 1 length for Mitred distributors. It is not needed for Full End of Diagonal distributors.
Nozzle DIameter	The internal diameter of the inlet/outlet nozzle. If omitted the inlet/outlet nozzle pressure loss is not calculated.

- Redistributor Type

Field	Description
Type	Allows you to specify the redistributor type, and the side of the exchanger on which the associated header is located. You have four options: • Standard • Twin • Hardway • Hardway Twin
Header Location	Allows you to specify the side of the exchanger that the header is located. You have three options: • Right side • Left side • Twin
Distance to Redistributor	Allows you to specify the distance to the redistributor from the inlet.
Fin Number for Pad 1, 2, and 3	Numbers to identify the fins used in the redistributor. The same fin is usually used in all pads, so only pad 1 need normally be specified. In a dividing redistributor, flow that remains in the layers flows through Pad 1 then Pad 2, while Pad 3 carries the flow that goes to other layers.
Dimension a (axial length)	Dimension a, the length along the exchanger occupied by the redistributor.
Dimension b	Dimension b for the redistributor. In a conventional dividing redistributor, this is the entry width associated with the flow that remains in the same layer.

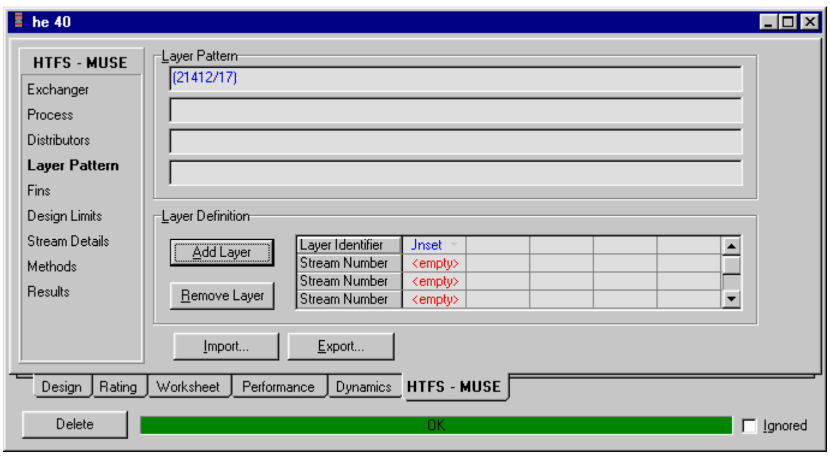
- Re-Inlet Distributor Type

Field	Description
Type	Allows you to specify the re-inlet distributor type. This can be in any form of side entry/exit distributor. You have five options: • None • Diagonal • Mitred • Indirect • Hardway
Fin Number for Pad 1 and 2	Numbers to identify the fins used in the re-inlet distributor pads. The same fin is usually used in both pads, so only pad 1 need normally be specified. Pad 1 is adjacent to the header.
Dimension a (axial length)	Dimension a, the length along the exchanger occupied by the re-inlet distributor.
Dimension b	Dimension b for the re-inlet distributor.
Extra Layers/ Draw Off Fraction	For a re-inlet distributor that directs flow to a number of extra layers, enter the number of extra layers. For a re-inlet distributor that collects from the extra layers, to direct it back to the basic number of layers, enter the number of extra layers with a minus sign. If there are no extra layers, but the stream is partially drawn off, enter the fraction of the stream that is drawn off.

Layer Pattern Page

The Layer Pattern page allows you to define the sequence of stream numbers that comprise the exchanger.

Figure 4.123



The layer pattern itself gives the sequence of layers, while the Layer Definition table lets you define the sequence of streams in each layer. The layer pattern is mandatory input for Layer-by-Layer simulations, but optional for stream by stream. If no layer pattern is provided, the number of layers for each stream must be specified.

Layer Pattern

Enter the sequence of layers forming the layer pattern (stacking pattern). The pattern can be identified as a sequence of layer identifiers, each identified by a letter, such as ABABABCAB. Though in simple cases, for example when there is only one stream per layer, the layer pattern can be specified as sequence of streams, for example 121213412.

Repeated sequences can be written in brackets, for example (121213/5)1312 means that the sequence 121213 occurs five times, followed by 1312. Spaces in the pattern are ignored, and brackets cannot be embedded in brackets. A stream number's sequence can contain zeros to indicate completely empty layers.

A layer pattern can terminate in M or MM to indicate that the pattern has central symmetry. MM indicates that the central layer is repeated, M that the symmetry is about the centre of the final layer. When The pattern is defined in terms of letters, use | or ||, not M's, to indicate mirror symmetry.

Layer Definition

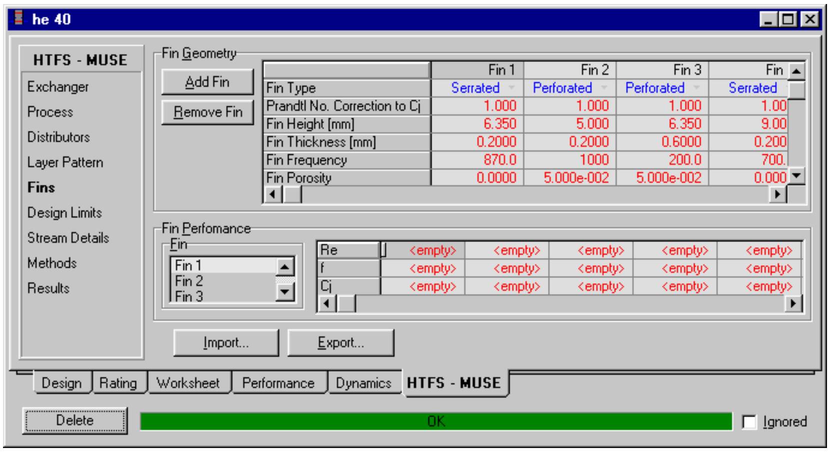
For each (alphabetic) layer identifier in the pattern specify the stream or sequence of streams along the exchanger from end A, within each layer type. This is only needed when a pattern is defined in terms of layer types (A, B, C, and so forth) rather than streams (1, 2, 4, and so forth).

> **The Layer Definition facility is only available in MUSE 3.20 and later versions.**

Fins Page

The Fins page allows you to specify data on fin geometry.

Figure 4.124



When a fin is specified, the corresponding fin performance data from the fin manufacturer (friction factors and Colburn j factors over a range of Reynolds numbers) should be input when available. If they are not available, they are estimated using generalized HTFS correlations for particular fin types.

Fin numbers are used to identify the particular fin used as main fin or distributor fin for each stream. Fin numbers up to 20 identify fins that data is specified in the program input.

The Fin Geometry groups consists of two buttons and a table. The two buttons allows you to add and remove fins from the heat exchanger, while the table allows you to specify each fin’s geometry.

The table consists of the following fields.

Field	Description
Fin Type	There are four main types of fin • Plain • Perforated • Serrated, or offset-strip • Wavy or herringbone For details of when each type should be used, refer to the MUSE Help file.
Prandtl No. Correlation to Cj	This parameter is important for high viscosity fluids in plain or perforated fins. The Colburn j factor assumes that Cj is a function of Re only, but this is not true at low Reynolds numbers (below 1000), where there is also Prandtl number dependence. If you specify the Re-f-Cj data at a Pr appropriate to the fluids used, omit this item. If you specify the Pr=1 data, specify 1 for a full correction, or a value between 0 and 1 for a partial correction. Refer to the MUSE Help file for more details.
Fin Height	Distance between the separating plates (parting sheets). This applies to all fin types. All the fins (main fin and distributor) for a stream must have the same height. A warning is issued if this is not so.
Fin Thickness	The fin thickness.
Fin Frequency	Number of fins per unit of length. This item can be zero if no fins are present. Common fin frequencies are 16, 18 or 21 ft/in for main fins, and 6 or 8 ft/in for distributor fins.
Fin Porosity	For perforated fins, enter the fin porosity as a fraction of the metal lost as holes.
Fin Serration Length	For serrated fins enter the fin serration length. The default is 3 mm (approximately 1/8 inch), which is typical of values used by most manufacturers. This input item is only needed for long-serration length serrated fins.

As mentioned above the corresponding fin performance data from the fin manufacturer, if available, should be entered into the Fin Performance group. To specify the data, select the fin number from the list of fins, and enter the data in the appropriate fields.

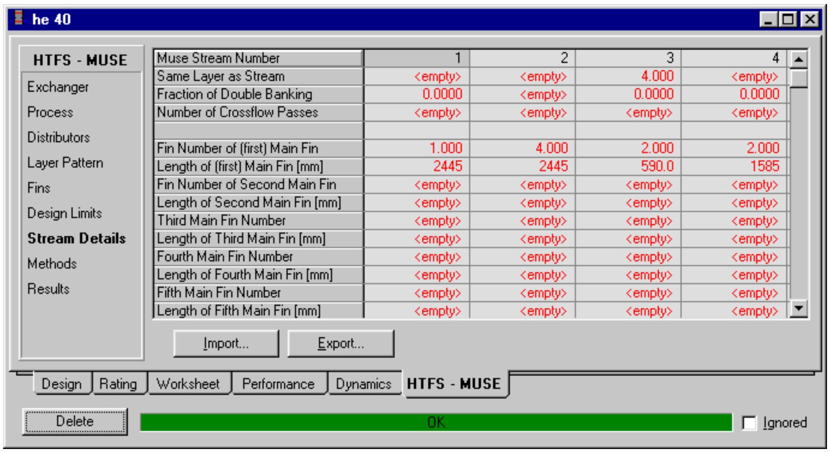
Design Limits Page

In the future version of HYSYS, the HTFS design capability will be available on the Design Limits page.

Stream Details Page

The Stream Details page allows you to specify more stream geometry data that supplements the data that was entered on the Exchanger page.

Figure 4.125



The following table defines each of the fields on this page.

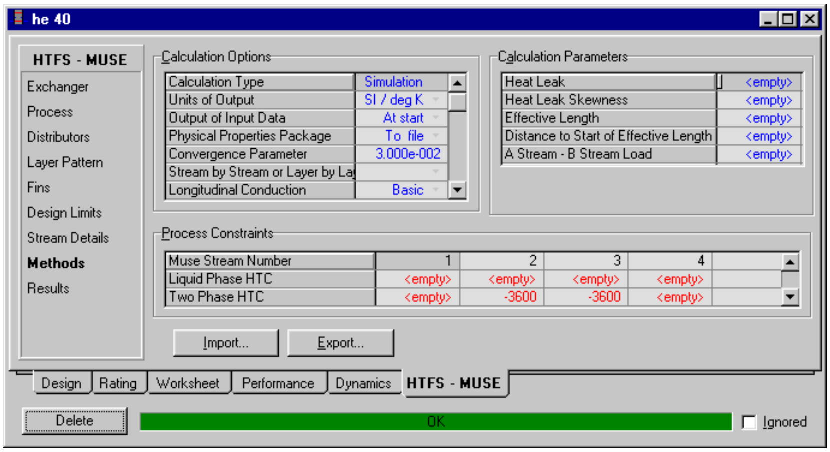
Field	Description
Same Layer as Stream	The Same Layer as Stream parameter is one way of specifying that two streams occupy the same set of layers in an exchanger. It is not needed, if you specify a layer pattern in terms of Layer Identifiers A, B, C, and so forth, with Layer Definition information. If you specify a layer pattern in terms of stream numbers, then one stream in each layer is used to identify that layer. For other streams in that layer give the number of the stream in the layer pattern that identifies the layer.
Fraction of Double Banking	The Fraction of Double Banking parameter is not needed if you specify a layer pattern, and is estimated if you do not. Refer to the MUSE Help for a definition and more details.
Number of Cross Flow Passes	For streams in multipass crossflow, enter the number of crossflow passes.
Fin Number of (first) Main Fin	Number to identify the main heat transfer fin for the stream. The number must correspond to one of the fin data blocks in the Fins page, or to a fin in a User Databank. If the stream uses more than one type of main fin, this item is the first fin, counting from the stream inlet.
Length of (first) Main Fin	The length of the main fin for the stream.
Fin Number for Second to Sixth Fins	For a stream that uses more than one type of main fin, enter the fin numbers. The sequence is from stream inlet to outlet.
Length of Fins for Second to Sixth Fins	The length of each main fin for the stream.

Methods Page

The Methods page consists of three groups:

- Calculation Options
- Calculation Parameters
- Process Constraints

Figure 4.126



Calculation Options Group

The Calculation Options group is used to configure the data that appears on the Results page. Only the Output Units need normally be set, as defaults are usually acceptable for all other items.

The group consists of the following fields.

Field	Description
Calculation Type	The Calculation Type should not be set. It gives access to a deprecated calculation facility, Length estimation, as an alternative to Normal simulation. Refer to the MUSE help file for more information.
Units of Output	Allows you to specify the set of units you want to use for the output data. There are five options to choose from: • SI / deg C • British • Metric / C • SI / deg K • Metric / K
Output of Input Data	Specifies where the output of input data appears in the main Results page (lineprinter output). Refer to the MUSE help for more information.
Physical Properties Package	Allows you to send the Physical Properties of the exchanger to the Results page or to a specific file.
Convergence Parameter	Use only if MUSE shows convergence problems. Refer to the MUSE help for more information.
Stream by Stream or Layer by Layer	Normally leave this item unset. Use it only with the (MULE) layer-by-layer calculation engine to enforce a reduced stream-by-stream calculation. Refer to the MUSE help for more information.
Longitudinal Conduction	You can specify that in addition to heat transfer between streams, allowance can be made for heat conducted in the exchanger metal from the hot to cold end of the exchanger. This can be important for exchangers used in liquefying hydrogen or helium. Refer to the MUSE help for more information.
Number of Calculation Steps	Calculations in MUSE use a number of equal length steps along the exchanger. The default is 100, the maximum 200. Refer to the MUSE help file for more information.
1st Est. Heat Load (fraction of max)	Use this item only if there are convergence problems. It lets you change the initial estimate of exchanger duty. Refer to the MUSE help file for more information.
Distributor Calculations	Gives you control over when distributor pressure losses are calculated. The default is that losses are calculated if you specify distributor data. Refer to the MUSE help for more information.
Maximum Number of Iterations	Enter a value if you want to restrict the number of iterations. Refer to the MUSE help file for more information.

Calculation Parameters Group

The Calculation Parameters group allows you to specify process exchanger parameters. The group consists of the following fields:

Field	Description
Heat Leak	It is possible to specify a net heat leak into the exchanger, or out of the exchanger, if a negative value is specified.
Heat Leak Skewness	For heat leaks that are not uniform along the exchanger length. Refer to the MUSE help for more information.
Effective Length, Distance to Effective Length	These two fields should normally be omitted, and left to the program to calculate. The effective length is that region of the exchanger where heat transfer is assumed to occur. It is determined from the exchanger geometry data, but you can override the program if you want. Refer to the MUSE help for more information.
A Stream - B Stream Load	A deprecated input. It is possible to specify the heat load across the exchanger for end A to end B.

Process Constraints Group

The Process Constraints group allows you to specify sets of stream constraints for over-riding, or scaling values normally calculated by the program. These should not be used, unless you have a good reason for doing so.

Field	Description
Liquid Phase HTC	You can enter a value for the liquid heat transfer coefficient here to override the calculated value. It is recommended that the program calculated values be used.
Two Phase HTC	You can enter a value for the two phase (boiling or condensing) heat transfer coefficient here to override the calculated value. It is recommended that the program calculated values be used.
Vapour Phase HTC	You can enter a value for the vapour heat transfer coefficient here to override the calculated value. It is recommended that the program calculated values be used.

Field	Description
Multiplier for Liquid Coefficient	A value entered here can be used to increase or decrease the calculated liquid heat transfer coefficient. It also scales any pre-set coefficient you input. It is recommended that the program calculated values be used.
Multiplier for Two Phase Coefficient	A value entered here can be used to increase or decrease the calculated boiling or condensing heat transfer coefficient. It also scales any pre-set coefficient you input. It is recommended that the program calculated values be used.
Multiplier for Vapour Coefficient	A value entered here can be used to increase or decrease the calculated vapour or gas heat transfer coefficient. It also scales any pre-set coefficient you input. It is recommended that the program calculated values be used.
Pressure Drop Multiplier	Enter the number that the calculated frictional pressure gradient (liquid, two phase or vapour) should be multiplied. It is not possible to scale the pressure drops of each phase separately. It is recommended that the program calculated values be used.
Precalculated Arrays Flag	Allows you to override an internal calculation flag, it is best left unset. See the MUSE help for more details.
Preset deltaT for Boiling	Provides a variant on the boiling method, it is best left unset. See the MUSE help for more details.

Results Page

The exchanger results appear on this page. The results are created in a text format that can be exported to HTFS-MUSE.

4.6 References

[1] Perry, R.H. and D.W. Green. Perry's Chemical Engineers' Handbook (Seventh Edition) McGraw-Hill (1997) p. 11-33

[2] Perry, R.H. and D.W. Green. Perry's Chemical Engineers' Handbook (Seventh Edition) McGraw-Hill (1997) p. 11-42

[3] Kern, Donald Q. Process Heat Transfer McGraw-Hill International Editions: Chemical Engineering Series, Singapore (1965) p. 139

5 Logical Operations

5.1 Adjust

The Adjust operation varies the value of one stream variable (the independent variable) to meet a required value or specification (the dependent variable) in another stream or operation.

The Adjust is a steady state operation; HYSYS ignores it in dynamic mode.

In a flowsheet, a certain combination of specifications may be required, which cannot be solved directly. These types of problems must be solved using trial-and-error techniques. To quickly solve flowsheet problems that fall into this category, the Adjust operation can be used to automatically conduct the trial-and-error iterations for you.

The Adjust is extremely flexible. It allows you to link stream variables in the flowsheet in ways that are not possible using ordinary “physical” unit operations. It can be used to solve for the desired value of just a single dependent variable, or multiple Adjusts can be installed to solve for the desired values of several variables simultaneously.

The Independent variable cannot be a calculated value; it must be specified.

The Adjust can perform the following functions:

- Adjust the independent variable until the dependent variable meets the target value.
- Adjust the independent variable until the dependent variable equals the value of the same variable for another object, plus an optional offset.

5.1.1 Adjust Property View

There are two ways that you can add an Adjust to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.

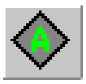
2. Click the **Logicals** radio button.
3. From the list of available unit operations, select **Adjust**.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing the **F4**.

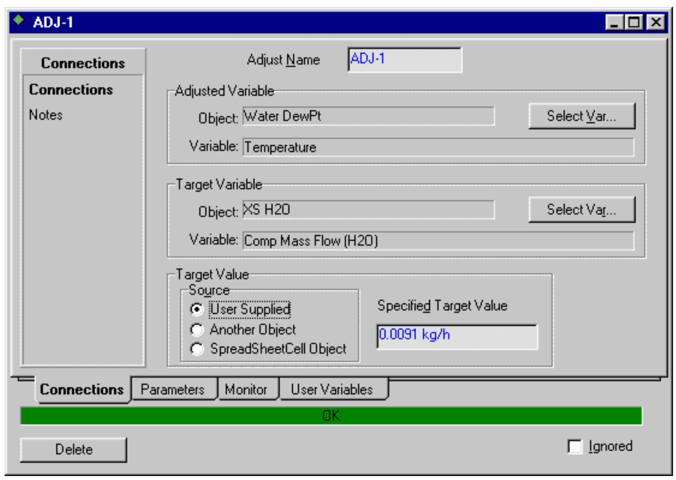
2. Double-click the **Adjust** icon.



Adjust icon

The Adjust property view appears.

Figure 5.1



5.1.2 Connections Tab

The first tab of the Adjust property view, as well as several other logicals, is the Connections tab. The tab contains the following pages:

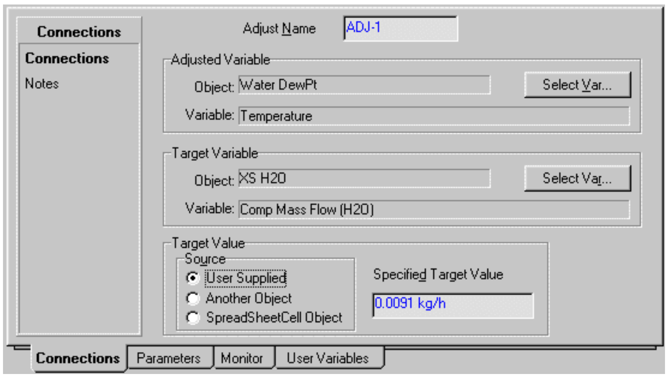
- Connections
- Notes

Connections Page

The Connections page comprises of three groups:

- Adjusted Variable
- Target Variable
- Target Value

Figure 5.2



Adjusted/Target Variable Groups

The Adjusted and Target Variable groups are very similar in appearance, each containing an Object field, Variable field, and a **Select Var** button.

- The Adjusted Object is the owner of the independent variable which is manipulated in order to meet the specified value of the **Target** variable.
- The Target Object is the owner of the dependent variable whose value you are trying to meet. A Target Object can be a unit operation, stream, or a utility.

Refer to **Section 1.3.9 - Variable Navigator Property View** for information.

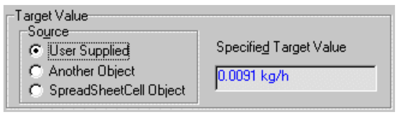
- The **Select Var** button enables you to select a variable for the Adjusted and Target objects.

Target Value Group

Once the target object and variable are defined, there are three choices for how the target is to be satisfied:

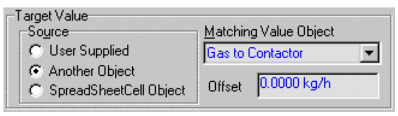
- If the target variable is to meet a certain numerical value, select the **User Supplied** radio button (as shown in the figure below), and enter the appropriate value in the Specified Target Value field.

Figure 5.3



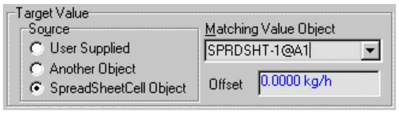
- If the target variable is to meet the value (or the value plus an offset) of the same variable in another stream or operation, select the **Another Object** radio button (as shown in the figure below), and select the stream or operation of interest from the Matching Value Object drop-down list. If applicable, enter an offset in the available field.

Figure 5.4



- If the target variable is to meet the value (or the value plus an offset) of the same variable specified in the spreadsheet, select the **SpreadSheetCell Object** radio button (as shown in the figure below), and select the cell that you want from the Matching Value Object drop-down list. This allows the SpreadSheetCell to be attached as an adjusted variable, and source to the target variable. You can also specify the offset in the available field.

Figure 5.5



Notes Page

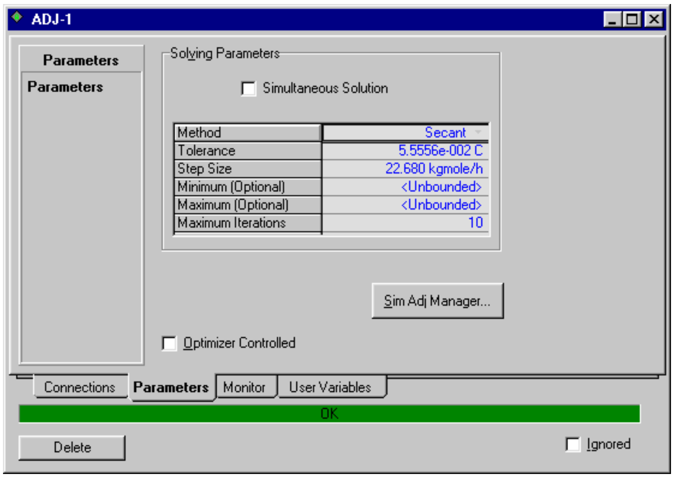
For more information, refer to **Section 1.3.5 - Notes Page/Tab**.

The Notes page provides a text editor, where you can record any comments or information regarding the operation or to your simulation case in general.

5.1.3 Parameters Tab

Once you have chosen the dependent and independent variables, the convergence criteria must be defined. This is usually done on the Parameters tab.

Figure 5.6



Solving Parameter	Description
Simultaneous Solution	Solves multiple Adjust loops simultaneously. There is only one simultaneous **solving** method available therefore when this checkbox is selected the **Method** field is no longer visible.
Method	Sets the (non-simultaneous) solving method: Secant or Broyden.
Tolerance	Sets the absolute error. In other words, the maximum difference between the Target Variable and the Target Value.
Step Size	The initial step size employed until the solution is bracketed.
Maximum / Minimum	The upper and lower bounds for the independent variable (optional) are set in this field.
Maximum Iterations	The number of iterations before HYSYS quits calculations, assuming a solution has not been obtained.
Sim Adj Manager	Opens the Simultaneous Adjust Manager allowing you to monitor and modify all Adjusts that are selected as simultaneous.
Optimizer Controlled	Passes a variable and a constant to the optimizer. When activated the efficiency of the simultaneous Adjust is increased. This option requires RTO.

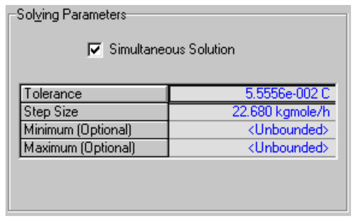
Choosing the Solving Methods

Adjust loops can be solved either individually or simultaneously. If the loop is solved individually, you have the choice of either a Secant (slow and sure) or Broyden (fast but not as reliable) search algorithm. The Simultaneous solution method uses modified Levenberg-Marquardt method search algorithm. A single Adjust loop cannot be solved in the Simultaneous mode. In Simultaneous mode, the adjust variable is adjusted after the last operation in the flowsheet has solved. The calculation level has no effect on the Adjust operation in the Simultaneous mode.

The Calculation Level for an Adjust (accessed under Main Properties) is 3500, compared to 500 for most streams and operations. This means that the Adjust is solved last among unknown operations. You can set the relative solving order of the Adjusts by modifying the Calculation Level.

When the **Simultaneous Solution** checkbox is selected, the **Method** field is no longer visible.

Figure 5.7



Simultaneous Adjust Manager

The Simultaneous Adjust Manager (SAM) property view allows you to monitor, and modify all Adjusts that are selected as simultaneous. This gives you access to a more efficient method of calculation, and more control over the calculations.

All adjusts from old cases in Simultaneous mode are automatically added to the SAM.

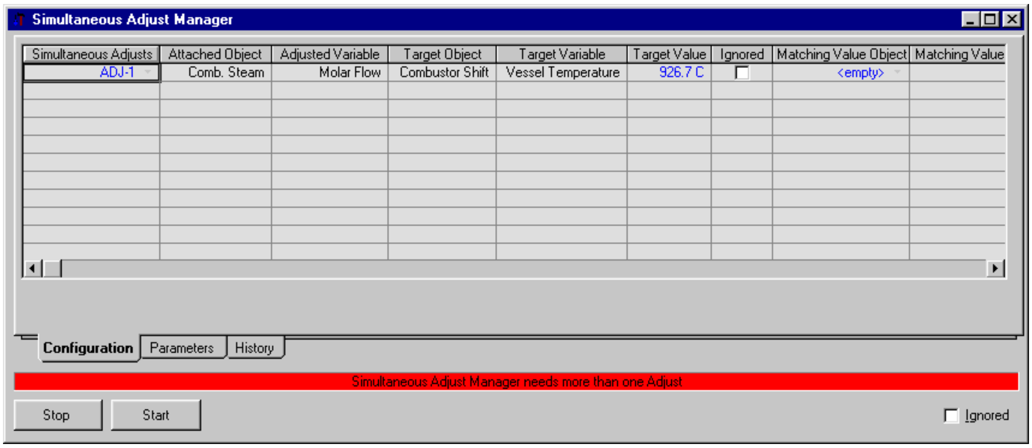
The SAM property view is launched by clicking the **Sim Adj Manager** button on the **Parameters** tab, or by selecting **Simultaneous Adjust Manager** command from the **Simulation** menu.

The SAM requires two or more active (in other words, not ignored) adjusts to solve. If you are using only one adjust, you cannot use the SAM.

The SAM property view contains the following tabs:

- Configuration
- Parameters
- History

Figure 5.8



The SAM property view also contains the following common objects at the bottom of the property view:

- The status bar, which displays the status of the SAM calculation.
- The **Stop** and **Start** buttons, which are used to start and stop the SAM calculations repectively.
- The **Ignored** checkbox, which enables you to toggle on and off the SAM feature and all of the selected Adjusts simultaneously.

Configuration Tab

The Configuration tab displays information regarding Adjusts that have been selected as simultaneous. You can view the individual Adjusts by double-clicking on the Adjust name. You can also modify the target value or matching value object, value, and offset. This tab also allows you to ignore individual Adjusts.

Parameters Tab

The Parameters tab allows you to modify the tolerance, step size, max, and min values for each Adjust, as well as, 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 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 (sums of squares) is less then a user specified value. • **Either**. SAM is converged with which ever 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.
Perturbation 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.

History Tab

The History tab displays the target value, adjusted value, and residual value for each iteration of the selected Adjust(s). One or more Adjusts can be displayed by clicking on the checkbox beside the Adjust name. The Adjusts are always viewed in order from left to right across the page. For example, if you are viewing Adjust 2 and add Adjust 1 to the property view, Adjust 1 becomes the first set of numbers, and Adjust 2 is shifted to the right.

The History tab only displays the values from a solve step. The values calculated during a Jacobian step can be seen on the Monitor tab of the adjust for the individual results.

Tolerance

For the Adjust to converge, the error in the dependent variable must be less than the Tolerance.

$$Error = |Dependent\ Variable\ Value - Target\ Value| \tag{5.1}$$

It is sometimes a good idea to use a relatively loose (large) tolerance when initially attempting to solve an Adjust loop. Once you determine that everything is working properly, you can reset the tolerance to the final design value.

The tolerance and error values are absolute (with the same units as the dependent variable) rather than relative or percentage-type.

Step Size

The step size you enter is used by the search algorithm to establish the maximum step size adjustment applied to the independent variable. This value is used until the solution has been bracketed, at which time a different convergence algorithm is applied. The value which is specified should be large enough to permit the solution area to be reached rapidly, but not so large as to result in an unreasonable overshoot into an infeasible region.

A positive step size initially increments the independent variable, while a negative value initially decrements the independent variable.

A negative initial step size causes the first step to decrement the independent variable.

If the Adjust steps away from the solution, the direction of the steps are automatically reversed.

Before installing the Adjust module, it is often good practice to initialize the independent variable, and perform one adjust "manually". Solve your flowsheet once, and notice the value for the dependent variable, then self-adjust the independent variable and re-solve the flowsheet. This assures you that one variable actually affects the other, and also gives you a feel for the step size you need to specify.

Maximum/Minimum

These two optional criteria are the allowable upper and lower bounds for the independent variable. If either bound is encountered, the Adjust stops its search at that point.

The Independent variable must be initialized (have a starting value) in order for the Adjust to work.

Maximum Iterations

The default maximum number of iterations is 10. Should the Adjust reach this many iterations before converging, the calculations stop, and you are asked if you want to continue with more iterations. You can enter any value for the number of maximum iterations.

5.1.4 Monitor Tab

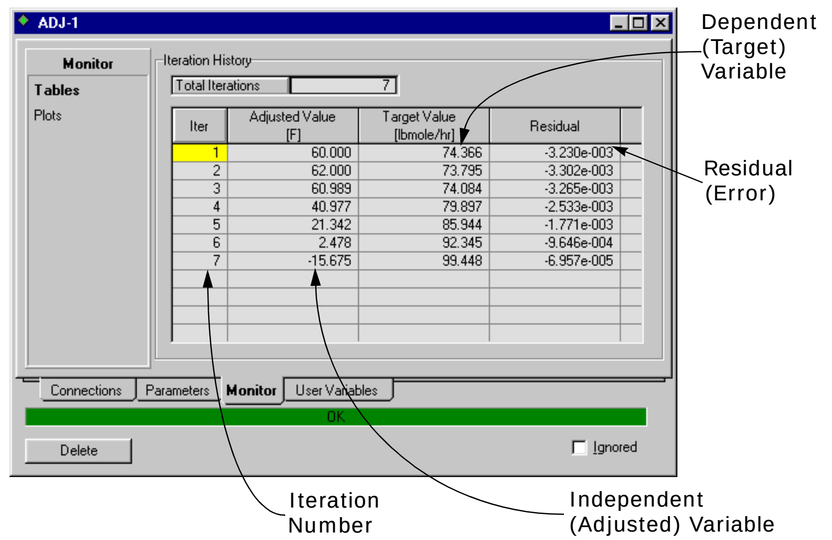
The Monitor tab contains the following pages:

- Tables
- Plots

Tables Page

For each Iteration of the Adjust, the number, adjusted value, target value, and residual appear. If necessary, use the scroll bar to view iterations which are not currently visible.

Figure 5.9



You can also use the Solver Trace Window to view the Iteration History.

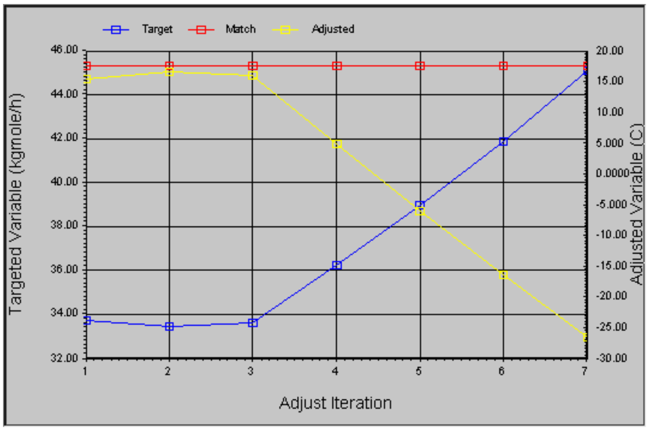
Refer to **Section 1.3 - Object Status & Trace Windows** in the **HYSYS User Guide** for more information.

Plots Page

Refer to **Section 1.3.1 - Graph Control Property View** for information on customizing plots.

The Plots page displays the target and adjusted variables like on the Tables page, except the information is presented in graph form.

Figure 5.10



5.1.5 User Variables Tab

For more information, refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables tab enables you to create and implement your own user variables for the current operation.

5.1.6 Starting the Adjust

There are two ways to start the Adjust:

- If you have provided values for all the fields on the **Parameters** tab, the Adjust automatically begins its calculations.
- If you have omitted one or both values in the Minimum/ Maximum fields (on the **Parameters** tab) for the independent variable (which are optional parameters), and you would like the Adjust to start calculating, simply click the **Start** button.

With the exception of the minimum and maximum values of the independent (adjusted) variable, all parameters are required before the Adjust begins its calculations.

The Start button then disappears, indicating the progress of the calculations. When the error is less than the tolerance, the status bar displays in green the "OK" message. If the Adjust reaches the maximum number of iterations without converging, the "Reached iteration Limit without converging" message appears in red on the status bar.

If you click the Start button when all of the required parameters are not defined, the status bar displays in yellow the "Incomplete" message, and calculations cannot begin.

Once calculations are underway, you can view the progress of the convergence process on the Iterations tab.

The Start button only appears in the initialization stage of the Adjust operation. It disappears from the property view as soon as it is pressed. Any changes made to the Adjust or other parts the flowsheet automatically triggers the Adjust calculation.

To stop or disable the Adjust select the Ignored checkbox.

5.1.7 Individual Adjust

The Individual Adjust algorithm, either Secant or Broyden, uses a step-wise trial-and-error method, and displays values for the dependent and independent variables on each trial. The step size specified on the Parameters tab is used to increment, or decrement the independent variable for its initial step. The algorithm continues to use steps of this size until the solution is bracketed. At this point, depending on your choice, the algorithm uses either the Secant search (and its own step sizes) or Broyden search to quickly converge to the desired value. If a solution has not been reached in the maximum number of iterations, the routine pauses, and asks you whether another series of trials should be attempted. This is repeated until either a solution is reached, or you abandon the search. The Secant search algorithm generally results in good convergence once the solution has been bracketed.

5.1.8 Multiple Adjust

The term Multiple Adjust typically applies to the situation where all of the Adjusts are to be solved simultaneously. In this case, where the results of one Adjust directly affect the other(s), you can use the Simultaneous option to minimize the number of flowsheet iterations.

Refer to **Chapter C2 - Synthesis Gas Production** in the **HYSYS Tutorials and Applications** guide for an example using Multiple Adjusts.

Examples where this feature is very valuable include calculating the flow distribution of pipeline looping networks, or in solving a complex network of UA-constrained heat exchangers. In these examples, you must select the stream parameters which HYSYS is to manipulate to meet the desired specifications. For a pipeline looping problem, the solution may be found by adjusting the flows in the branched streams until the correct pressures are achieved in the pipelines downstream. In any event, it is up to you to select the variables to adjust to solve your flowsheet problem.

HYSYS uses the modified Levenberg-Marquardt algorithm to simultaneously vary all of the adjustable parameters defined in the Adjusts until the desired specifications are met. The role of step size with this method is quite different. With the single Adjust algorithm, step size is a fixed value used to successively adjust the independent variable until the solution has been bracketed. With the simultaneous algorithm, the step size for each variable serves as an upper limit for the adjustment of that variable.

Refer to **Section 4.4 - Heat Exchanger** for more information.

In solving multiple UA exchangers, the starting point should not be one that contains a temperature crossover for one of the heat exchangers. If this occurs, a warning message appears informing you that a temperature crossover exists, and a very large UA value is computed for that heat exchanger. This value is insensitive to any initial change in the value of the adjustable variable, and therefore the matrix cannot be solved.

One requirement in implementing the Multiple Adjust feature is that you must start from a feasible solution.

Install all Adjusts using the simultaneous option on the Parameters tab, then click the Start button to begin the calculations.

5.2 Balance

The Balance operation provides a general-purpose heat and material balance facility. The only information required by the Balance is the names of the streams entering and leaving the operation. For the General Balance, component ratios can also be specified.

Since HYSYS permits streams to enter or leave more than one operation, the Balance can be used in parallel with other units for overall material and energy balances.

The Balance overrides the filtering of streams that HYSYS typically performs.

The Balance Operation solves in both the forward and backward directions. For instance, it backs out the flowrate of an unknown feed, given that there are no degrees of freedom.

There are six Balance types which are defined in the table below:

Balance Type	Definition
Mole	An overall balance is performed where only the molar flow of each component is conserved. It can be used to provide material balance envelopes in the flowsheet, or to transfer the flow and composition of a process stream into a second stream.
Mass	An overall balance is performed where only the mass flow is conserved. A common application would be for modeling reactors with no known stoichiometry, but for which analyses of all feeds and products are known.
Heat	An overall balance is performed where only the heat flow is conserved. An application would be to provide the pure energy difference in a heat balance envelope.
Mole and Heat	An overall balance is performed where the heat and molar flow are conserved. The most common application for this unit operation would be to perform overall material (molar basis) and energy balance calculations of selected process streams to either check for balances, or force HYSYS to calculate an unknown variable, such as flow. Most of the unit operations in HYSYS perform the equivalent of a Mole and Heat Balance besides their other more specialized calculations.
Mass and Heat	An overall balance is performed where the overall mass flow and heat flow are conserved.
General	HYSYS solves a set of n unknowns in the n equations developed from the streams attached to the operation. Component ratios can be specified on a mole, mass or liquid volume basis.

5.2.1 Balance Property View

There are two ways that you can add a Balance to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

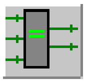
   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Logicals** radio button.
3. From the list of available unit operations, select **Balance**.

4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.

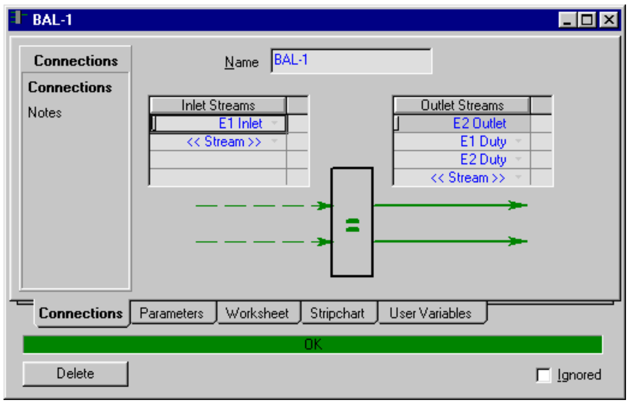


Balance icon

2. Double-click the **Balance** icon.

The Balance property view appears.

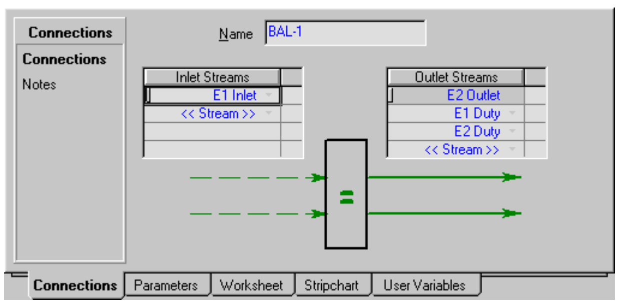
Figure 5.11



5.2.2 Connections Tab

The Connections tab is the same for all of the Balance Types.

Figure 5.12



The tab contains the following pages:

- Connections
- Notes

Refer to **Section 1.3.5 - Notes Page / Tab** for more information.

Connections Page

On the Connections page, you must specify the following information:

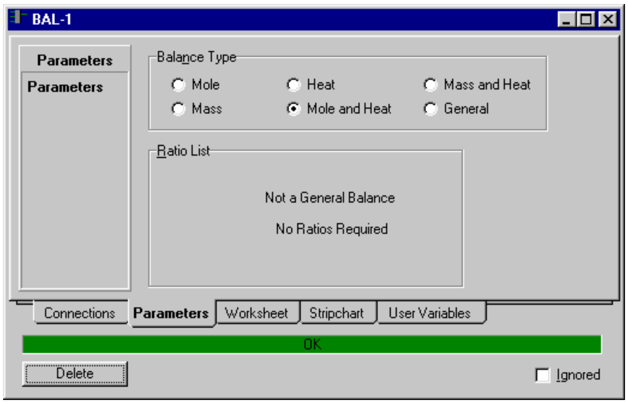
- **Name**. The name of the balance operation.
- **Inlet Streams**. Attach the inlet streams to the balance.
- **Outlet Streams**. Enter the outlet streams to the balance operation. You can have an unlimited number of inlet and outlet streams. Use the scroll bar to view streams that are not visible.

5.2.3 Parameters Tab

The Parameters tab contains two groups:

- Balance Type
- Ratio List

Figure 5.13



The Balance Type group contains a series of radio buttons, which allow you to choose the type of Balance you want to use. The radio buttons are:

- Mole
- Mass

- Heat
- Mole and Heat
- Mass and Heat
- General

The Ratio List group applies only to the General balance. This is discussed in the General Balance section.

Mole Balance

This operation performs an overall mole balance on selected streams; no energy balance is made. It can be used to provide material balance envelopes in the flowsheet or to transfer the flow and composition of a process stream into a second stream.

- The composition does not need to be specified for all streams.
- The direction of flow of the unknown is of no consequence. HYSYS calculates the molar flow of a feed to the operation based on the known products, or vice versa.
- This operation does not pass pressure or temperature.

Mass Balance

This operation performs an overall balance where only the mass flow is conserved. An application is the modeling of reactors with no known stoichiometry, but for which analyses of all feeds and products are available. If you specify the composition of all streams, and the flowrate for all but one of the attached streams, the Mass Balance operation determines the flowrate of the unknown stream. This is a common application in alkylation units, hydrotreaters, and other non-stoichiometric reactors.

- The composition must be specified for all streams.
- The flowrate must be specified for all but one of the streams. HYSYS determines the flow of that stream by a mass balance.
- Energy, moles, and chemical species are not conserved. The Mass Balance operation determines the equivalent masses of the components you have defined for the inlet and outlet streams of the operation.
- This operation does not pass pressure or temperature.

Heat Balance

This operation performs an overall heat balance on selected streams. It can be used to provide heat balance envelopes in the flowsheet or to transfer the enthalpy of a process stream into a second energy stream.

- The composition and material flowrate must be specified for all material streams. The heat flow is not passed to streams which do not have the composition and material flowrate specified, even if there is only one unknown heat flow.
- The direction of flow for the unknown stream is of no consequence. HYSYS calculates the heat flow of a feed to the operation based on the known products, or vice versa.
- This operation does not pass the pressure or temperature.
- You cannot balance the heat into a Material Stream.

Mole and Heat Balance

The most common application for this balance is to perform overall material (molar basis), and energy balance calculations of selected process streams to either check for balances or force HYSYS to calculate an unknown variable, such as a flowrate.

- The Mole and Heat Balance independently balance energy and material.
- The Mole and Heat Balance calculate ONE unknown based on a total energy balance, and ONE unknown based on a total material balance.
- The operation is not directionally dependent for its calculations. Information can be determined about either a feed or product stream.
- The balance remains a part of your flowsheet and as such defines a constraint; whenever any change is made, the streams attached to the balance always balances with regard to material and energy. As such, this constraint reduces by one the number of variables available for specification.
- Since the Mole and Heat Balance work on a molar basis, it should not be used in conjunction with a reactor where chemical species are changing.

Mass and Heat Balance

Similar to the Mass balance mode, this balance mode performs a balance on the overall mass flow. In addition, however, energy is also conserved.

- The composition must be specified for all streams.
- Flow rate must be specified for all but one of the streams. HYSYS determines the flow of that stream by a mass balance.
- Enthalpy must be specified for all but one of the streams. HYSYS determines the enthalpy of that stream by a heat balance.
- Moles and chemical species are not conserved.

General Balance

The General Balance is capable of solving a greater scope of problems. It solves a set of n unknowns in the n equations developed from the streams attached to the operation. This operation, because of the method of solution, is extremely powerful in the types of problems that it can solve. Not only can it solve unknown flows and compositions in the attached streams (either inlet or outlet can have unknowns), but ratios can be established between components in streams. When the operation determines the solution, the prescribed ratio between components are maintained.

- The General balance solves material and energy balances independently. An Energy Stream is an acceptable inlet or outlet stream.
- The operation solves unknown flows or compositions, and can have ratios specified between components in one of the streams.
- Ratios can be specified on a mole, mass or liquid volume basis.

Ratios

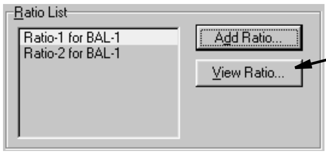
A Ratio, which is unique to the general Balance, is defined between two components in one of the attached streams. Multiple ratios within a stream (for example 1:2 and 1:1.5) can be set with a single Ratio on a mole, mass or liquid volume basis. Each individual ratio (1:2, 1:1, and 1:1.5), however uses a degree of freedom.

To set a ratio:

1. On the **Parameters** tab of the Balance operation property view, select the **General Balance** radio button.

   The Ratio List group should now be visible.

Figure 5.14

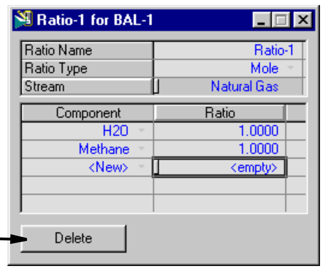


To view a specific ratio, select the ratio in the Ratio List group and click the **View Ratio** button.

2. Click the **Add Ratio** button to access the Ratio property view.

Figure 5.15

To delete a ratio, open the Ratio property view of the component ratio, and then click the **Delete** button in that Ratio property view.



3. In the Ratio property view, specify the following information:
   - **Name**. The name of the Ratio.
   - **Stream**. The name of the stream.
   - **Ratio Type**. Allows you to specify the Ratio Type: Mole, Mass, or Volume.
   - **Component/Ratio**. Provides the relative compositions of two or more components. Other components in the stream are calculated accordingly, and it is not necessary nor advantageous to include these in the table. All ratios must be positive; non-integer values are acceptable.

Number of Unknowns

The general Balance determines the maximum number of equations, and hence unknowns, in the following manner (notice that the material and energy balances are solved independently):

- One equation performing an overall molar flow balance.
- {Number of Components (*nc*)} equations performing an individual molar balance.
- {Number of Streams (*ns*)} equations, each performing a summation of individual component fractions on a stream by stream basis.

This is the maximum number of equations (1 + *nc* + *ns*), and hence unknowns, which can be solved for a system. When ratios are specified, they reduce the available number of unknowns. For each ratio, the number of unknowns used is one less than the number of components in the ratio. For example, for a three-component ratio, two unknowns are used.

5.2.4 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

5.2.5 Stripchart Tab

Refer to **Section 1.3.7 - Stripchart Page / Tab** for more information.

The Stripchart tab allows you to select and create default strip charts containing various variable associated to the operation.

5.2.6 User Variables Tab

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables tab enables you to create and implement your own user variables for the current operation.

5.3 Boolean Operations

The Boolean Logic block is a logical operation, which takes in a specified number of boolean inputs and then applies the boolean operation to calculate an output. A typical use of the Boolean Logic is to apply emergency shutdown of an exothermic reactor, such as closing the valves on the fuel and air line to the reactor when the reactor core temperature exceeds its setpoint. It is also used to simulate the ladder diagrams, which are found in most of the electrical applications.

The following Boolean Logic blocks are available in HYSYS:

- And Gate
- Or Gate
- Not Gate
- Xor Gate
- On Delay Gate
- Off Delay Gate
- Latch Gate
- Counter Up Gate
- Counter Down Gate
- Cause And Effect Matrix

For more information about the Integrator property view, refer to **Section 7.7 - Integrator** in the **HYSYS User Guide**.

To evaluate the Boolean Logic blocks at each time step, open the Integrator property view and go to the Options tab. In the Calculation Execution Rates group, change the Control and Logical Ops field value to 1.

This change ensures that your time sensitive Boolean Logic blocks like On Delay and Off Delay are executed at the required time instead of a one time step delay. This change also slows down the HYSYS calculation rate and is noticeable for large cases.

5.3.1 Boolean Logic Blocks Property View

There are two ways that you can add Boolean Logic Blocks to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

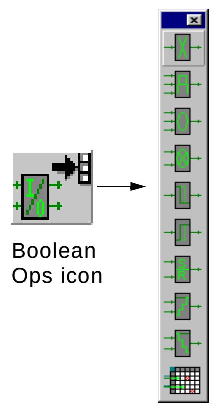
   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Logicals** radio button.
3. From the list of available unit operations, select the Boolean Logic that you want.
4. Click the **Add** button.

OR

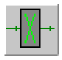
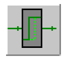
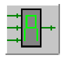
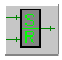
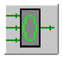
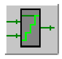
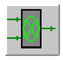
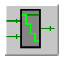
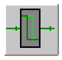
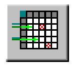
1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.
2. Click on the **Boolean Ops** icon. The Boolean Palette appears.

Figure 5.16

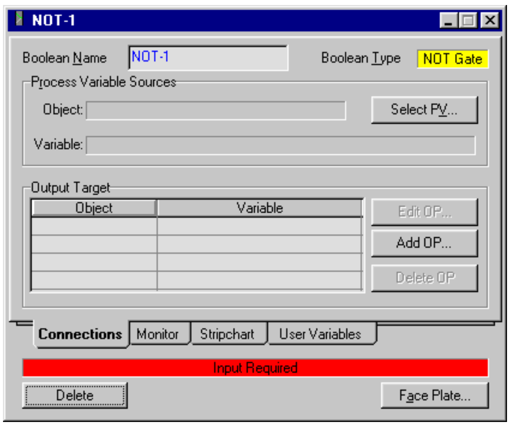


3. Double-click the icon of the Boolean Logic that you want.

Boolean Logic	Icon	Boolean Logic	Icon
Not Gate		**On Delay Gate**	
And Gate		**Latch Gate**	
Or Gate		**Counter Up Gate**	
Xor Gate		**Counter Down Gate**	
Off Delay Gate		**Cause And Effect Matrix**	

The selected Boolean Logic property view appears.

Figure 5.17

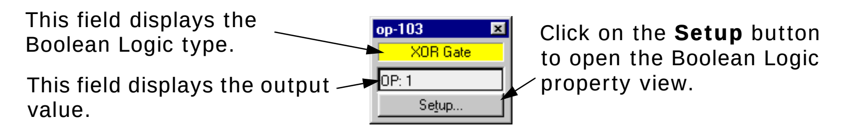


The property view for all the Boolean Logic blocks in HYSYS contains four tabs (Connections, Monitor, Stripchart, and User Variables), a Delete button, and a Face Plate button.

Logical Operation Face Plate Property View

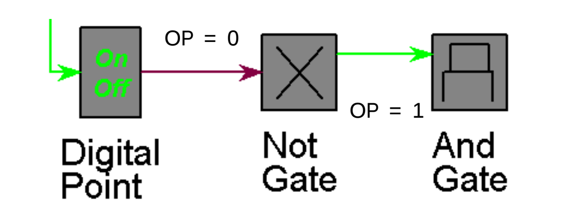
The **Face Plate** button enables you to access the Face Plate property view. The Face Plate property view allows you to see the Boolean type and output value at a glance.

Figure 5.18



On the PFD property view, the digital/boolean and boolean/ boolean logical connections have the capability to display the change of logical state by changing the line colour to either green (1) or red (0).

Figure 5.19



The output is set up to have a default initial value of 1 for all the Boolean Logic blocks.

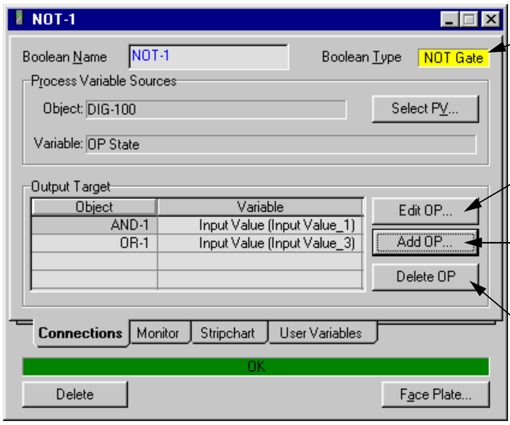
Connections Tab

The Connections tab is where you connect operations to the Boolean Logic block. Boolean unit operations can make logical connections with Digital Point, as well as, among themselves. The connections can either be made from the Connections tab, or through the PFD.

If the Boolean type supports multiple process variable sources, the Process Variable Sources group contains a table with three buttons with the same functions as the buttons in the Output Target group.

The figure below displays the Connections tab of a Boolean Not Gate operation.

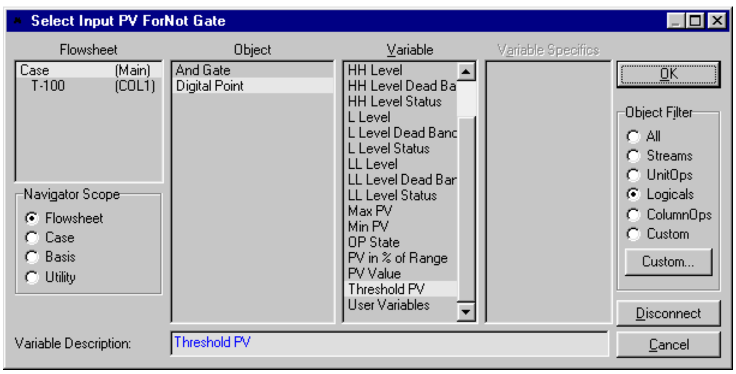
Figure 5.20



Adding/Editing Process Variable Source

Depending on the Boolean type, you have to click the Select PV button, the Edit PV button or the Add PV button to open the Select Input PV property view.

Figure 5.21



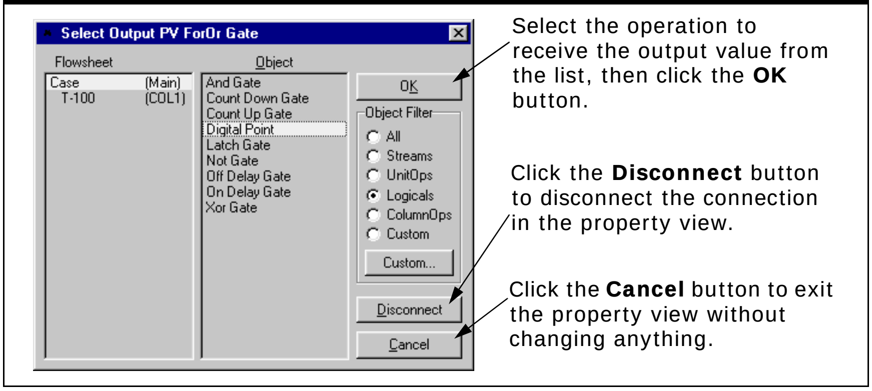
Refer to **Section 1.3.9 - Variable Navigator Property View** for information on.

The Select Input PV property view is similar to the Variable Navigator property view.

Adding/Editing Output Target

Click the Edit OP button or Add OP button to open the Select Output PV property view.

Figure 5.22



Use the radio button in the Object Filter group to filter the Object list to the operations you want.

Monitor Tab

The Monitor tab allows you to monitor the input and output values of the Boolean Logic block. The contents of this tab varies from one Boolean Logic type to another. For example, the Monitor tab of an On Delay Gate boolean also contains a field where you can specify the time delay.

Stripchart Tab

Refer to **Section 1.3.7 - Stripchart Page / Tab** for more information.

The Stripchart tab allows you to select and create default strip charts containing various variable associated to the operation.

User Variables Tab

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables tab enables you to create and implement your own user variables for the current operation.

5.3.2 And Gate

This unit operation performs a logical AND function on a set of inputs. The output is always low as long as any one of the input is low and it is high when all of the inputs are high. The table below displays the function logic for the And Gate.

Input 1	Input 2	Input 3	Output
1	1	1	1
1	0	0	0
0	1	0	0
0	0	1	0
0	0	0	0

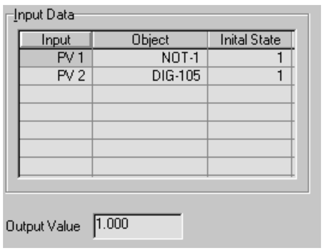
The And Boolean Logic block can have any number of inputs and a single output, which can be fanned out.

The input and output values can only be 1 or 0.

The Monitor tab of the And Gate displays the following information:

- **Input**. Contains the name and number used to designate the input connection.
- **Object**. Displays the operation name of the input connection.
- **Initial State**. Displays the input value received by the Boolean Logic block.
- **Output Value**. Displays the output value of the Boolean Logic block, based on the Boolean type and the input values from the input connections.

Figure 5.23



5.3.3 Or Gate

This unit operation performs a logical OR function on a set of inputs. The output is always high as long as any one of the input is high and it is low when all of the inputs are low. The table below displays the function logic for the Or Gate.

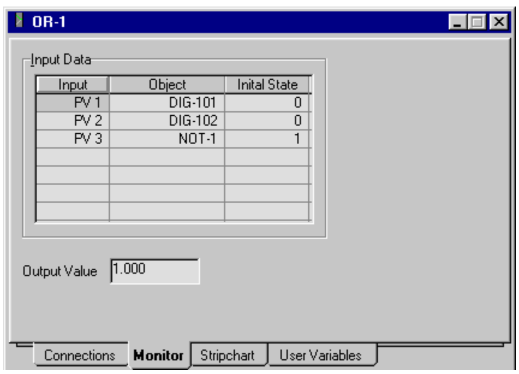
Input 1	Input 2	Input 3	Output
1	1	1	1
1	0	0	1
0	1	0	1
0	0	1	1
0	0	0	0

The Or Boolean Logic block can have any number of inputs and a single output, which can be fanned out.

The input and output values can only be 1 or 0.

The Monitor tab of the Or Gate displays the same information as the And Gate.

Figure 5.24



5.3.4 Not Gate

This unit operation perform a logical NOT function on an input. The output is the negative of the input. In other words, when the input is high the output is low and vice versa. The table below displays the function logic for the Not Gate.

Input	Output
1	0
0	1

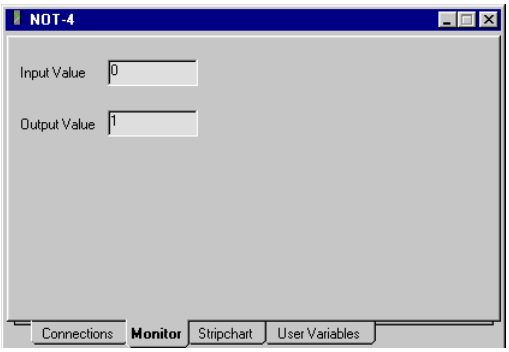
The input and output values can only be 1 or 0.

The output in this unit operation can also be fanned out.

The Monitor tab of the Not Gate displays the following information:

- **Input Value**. Displays the value received by the Boolean Logic block.
- **Output Value**. Displays the output value of the Boolean Logic block, based on the input value from the input connection.

Figure 5.25



5.3.5 Xor Gate

This unit operation performs an exclusive or function on two inputs. The output state is always High (1) whenever anyone of the input is high (1), but it is low (0) when all of the inputs are high (1). The table below displays the function logic for Xor Gate.

Input 1	Input 2	Output
1	1	0
1	0	1
0	1	1
0	0	0

The input and output values can only be 1 or 0.

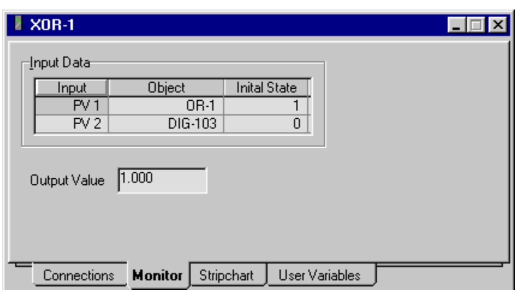
This unit operation can only have two input connections.

The output in this unit operation can also be fanned out.

The Monitor tab of the Xor Gate displays the following information:

- **Input**. Contains the name and number used to designate the input connection.
- **Object**. Displays the operation name of the input connection.
- **Initial State**. Displays the input value received by the Boolean Logic block.
- **Output Value**. Displays the output value of the Boolean Logic block, based on the Boolean type and the input values from the input connections.

Figure 5.26



5.3.6 On Delay Gate

This unit operation performs an on time delay function on a single input. The output's signal is delayed for a specified time delay (θ) only when the input is set to be 1. The following logical expression is used to calculate the output (y) for an input (x) change.

For $x = 1$,

$$y(t) = \begin{cases} 0 & t < \theta \\ 1 & t \geq \theta \end{cases} \qquad (5.2)$$

The Monitor tab of the On Delay Gate displays the following information:

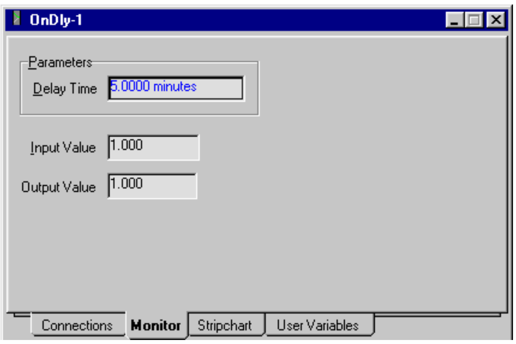
- **Delay Time**. Allows you to specify the amount of time you want for the time delay function. The default value is 10 minutes.
- **Input Value**. Displays the value received by the Boolean Logic block.

The input and output values can only be 1 or 0.

The output in this unit operation can also be fanned out.

- **Output Value**. Displays the output value of the Boolean Logic block, based on the input value from the input connection.

Figure 5.27



5.3.7 Off Delay Gate

This unit operation performs an off time delay function on a single input. The output's signal is delayed for a specified time delay (θ) only when the input is set to be 0. The following logical expression is used to calculate the output (*y*) for an input (*x*) change.

For $x = 0$,

$$y(t) = \begin{cases} 1 & t < \theta \\ 0 & t \geq \theta \end{cases} \tag{5.3}$$

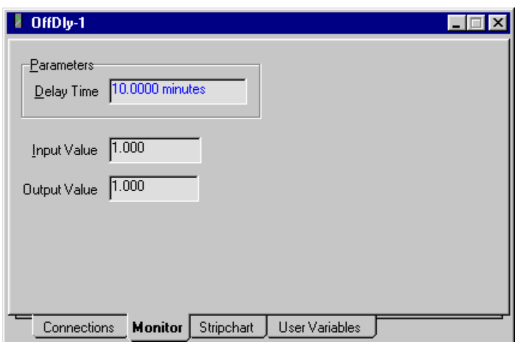
The Monitor tab of the Off Delay Gate displays the following information:

- **Delay Time**. Allows you to specify the amount of time you want for the time delay function. The default value is 10 minutes.
- **Input Value**. Displays the value received by the Boolean Logic block.

> **The input and output values can only be 1 or 0.**
>
> **The output in this unit operation can also be fanned out.**

- **Output Value**. Displays the output value of the Boolean Logic block, based on the input value from the input connection.

Figure 5.28



5.3.8 Latch Gate

This unit operation provides a latch functionality. It requires two input signals; one for set and other one for reset. The second input is the prevailing input meaning that it specify the output to be set to high (1), reset to low (0), or left unchanged. The table below displays the function logic for the Latch Gate.

Input 1	Input 2	Output
1	1	override state
1	0	1
0	1	0
0	0	previous state

By definition the latch gate allows you to select the OP value when both of its inputs are high. So this state is known in the industry as override state.

The Monitor tab of the Latch Gate displays the following information:

- **Prevailing Input**. The radio buttons come into play when both of the inputs are high(1). It allows you to specify what you want the output value to be. Selecting Set makes the OP value to be high(1), and Reset makes it low(0).
- **Input**. Contains the name and number used to designate the input connection.

The input and output values can only be 1 or 0.

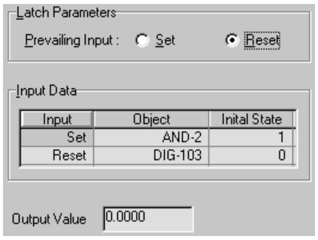
This unit operation can only have two input connections.

The output in this unit operation can also be fanned out.

- **Object**. Displays the operation name of the input connection.
- **Initial State**. Displays the input value received by the Boolean Logic block.

- **Output Value**. Displays the output value of the Boolean Logic block, based on the input value from the input connection.

Figure 5.29



5.3.9 Counter Up Gate

This unit operation acts as an up counter. It counts up to a maximum counter value which is specified by the users. It is triggered everytime the input is switched to a desired state. After reaching the maximum counter limit, it sets the output to a predefined value. The counter and output value is reset with the second input by switching it to high (1).

The Monitor tab of the Counter Up Gate displays the following information:

- **Maximum Counter**. Allows you to specify the counter limit value. The default value is 10.
- **Current Counter**. Displays the current counter value.
- **PV Alarm**. Allows you to select which PV value triggers the counter to increase a step. You can only choose 0 or 1.
- **Desired Output Value**. Allows you to select what the output value should be when the counter reaches maximum. You can only choose 0 or 1.
- **Input**. Contains the name and number used to designate the input connection.

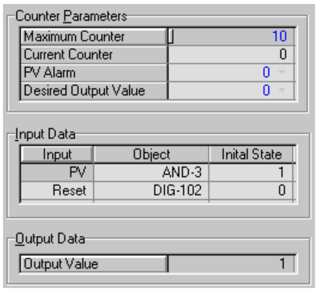
The input and output values can only be 1 or 0.

The output in this unit operation can also be fanned out.

- **Object**. Displays the operation name of the input connection.

- **Initial State**. Displays the input value received by the Boolean Logic block.
- **Output Value**. This field displays the output value of the Boolean Logic block, based on the input value from the input connection.

Figure 5.30



5.3.10 Counter Down Gate

This unit operation acts as a down counter. It counts down to a maximum counter value which is specified by the users. It is triggered everytime the input 1 is switched to a desired state. After the counter has reached zero, it sets the output to a predefined value. The counter and output value is reset with the second input by switching it to High (1).

The Monitor tab of the Counter Down Gate displays the following information:

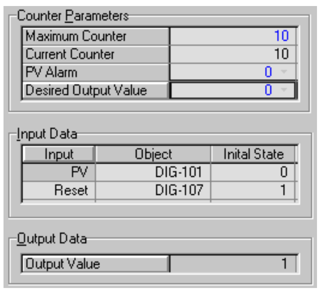
- **Maximum Counter**. Allows you to specify the counter limit value. The default value is 10.
- **Current Counter**. Displays the current counter value.
- **PV Alarm**. Allows you to select which PV value triggers the counter to decrease a step. You can only choose 0 or 1.
- **Desired Output Value**. Allows you to select what the output value should be when the counter reaches 0. You can only choose 0 or 1.

- **Input**. Contains the name and number used to designate the input connection.

> **The input and output values can only be 1 or 0.**
>
> **The output in this unit operation can also be fanned out.**

- **Object**. Displays the operation name of the input connection.
- **Initial State**. Displays the input value received by the Boolean Logic block.
- **Output Value**. Displays the output value of the Boolean Logic block, based on the input value from the input connection.

Figure 5.31



5.3.11 Cause and Effect Matrix

This unit operation replicates a Cause and Effect matrix commonly used in designing and operating the safety system of many processing plants. It looks at process values throughout the process and, based upon safety thresholds, determines if certain equipment and/or valves should be shutdown.

Refer to **Section 5.10 - Spreadsheet** for more information on the spreadsheet.

The unit operation is similar to a spreadsheet. It takes inputs called Causes, and sends outputs called Effects.

The input may be any simulation variable from the users case or a simple switch which is not required to be connected to an object’s variable. Each input generates either a Healthy (1) or Tripped (0) state.

The output is a boolean (1 or 0) result from processing one of the Cause and Effect Matrix columns. The output may write or export its result to any simulation variable within the users case. The user must specify a variable of discrete type (1 or 0). The output is not required to be connected to an object or variable. The same 1 or 0 result is produced from the matrix column, and then any other object in the simulation may read or use this value.

It is important that you clarify the 1 and 0 convention of the Cause and Effect Matrix for Healthy/Tripped, On/Off, Start/Stop, and so forth.

For both the inputs and outputs, a result of 0 indicates Tripped, whereas a result of 1 indicates Healthy, except where the Invert checkbox is turned on.

When the Invert checkbox is turned on, a result of 0 indicates Healthy, whereas a result of 1 indicates Tripped.

The matrix is processed one column at a time to determine the resultant state of the output associated with that column. The associated input state is reviewed for each element (or row entry) of a particular column having a non-blank user specified matrix element. All of the matrix elements of that column (and their associated input state) are compared based upon their respective and collective meaning to determine the Cause result.

You can access the Cause and Effect Matrix help property view by clicking on the Cause and Effect Help button on the C&E Matrix tab.

The boolean inputs enter through logical gate type operations (and, or, not, and so forth) with each other to determine the resultant boolean value.

Each matrix element type is described in the following table.

Matrix Elements		Description
X	TRIP	One or more zero input(s) causes a zero output.
R	RESET	One or more 1 inputs causes a 1 output (as long as there are no X, T or C active and ALL P must be 1) There is no requirement to have a reset on a particular output. If you want a reset, this can either be done with one or more R matrix element entries or a local reset switch. In the case of both R and a local reset, then both reset features must be reset for the output to return to normal, and the local reset must be done last.
T	TIMED TRIP	Same as the TRIP but the input must have remained zero for at least the time period. The T matrix element should be followed by an integer representing the number of seconds of time delayed trip.
C	COINCIDENT TRIP	In contrast to all other trips, a zero input for ALL the coincident signals of the same grouping causes a zero output. The C matrix element should be followed by an integer representing the Coincident group number. There should be more than one in each group.
P	PERMISSIVE	All P inputs must be 1 to permit an R to have the desired effect. Also required for a STANDBY 1 effect, a local reset and a local switch ON.
I	INHIBIT	A 1 will inhibit any trip of the output which would normally be caused by an X,T or C.
S	STANDBY	A 1 will cause a 1 (as long as there are no X, T or C active and ALL P must be 1), and a zero will cause a zero output (no INHIBIT applicable). Normally one would not want more than one Standby input designation per output. If you have more than one Standby, ALL Standby inputs must have a 1 for the output to be started (1 result). Otherwise, a zero output result is produced. All Permissive inputs must be 1 for the Standby 1 action to occur.

It is recommended that you build a dynamics case first with all the specifications in place before adding and configuring a Cause and Effect Matrix.

Configuring a Cause and Effect Matrix

Refer to **Section 5.10 - Spreadsheet** for more information on the spreadsheet.

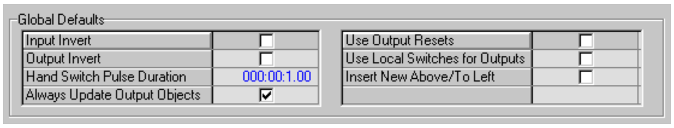
There are no PFD streams or lines that connect to or from the Cause and Effect Matrix operation. Hence you can place it anywhere and on any flowsheet. You can also view all simulation variables across flowsheets, the same as with a Spreadsheet.

To add a new Cause and Effect Matrix unit operation to the flowsheet, refer to **Section 5.3.1 - Boolean Logic Blocks Property View**.

You can set the global defaults and controls on the **Parameters** tab as shown in steps below:

1. You can specify the global defaults by clicking on the appropriate checkboxes.

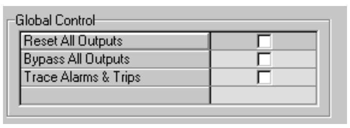
Figure 5.32



Checkbox	Description
Input Invert	Allows you to set the invert checkbox on for all new inputs.
Output Invert	Allows you to set the invert checkbox on for all new outputs.
Hand Switch Pulse Duration	You can specify the pulse duration for any hand Switch inputs that are pulsed.
Always Update Output Objects	You can select this checkbox, if you want to ensure that the results from the logic calculations are sent every timestep to the output objects.
Use Output Resets	Select this checkbox if you want the **Use Reset** checkbox selected for all new outputs.
Use Local Switches for Outputs	You can select this checkbox, if you want the **Use Local Switch** checkbox selected for all new outputs.
Insert New Above/To Left	When you add a new row or column and this checkbox is selected, the row or column will be added above (to the left) of that currently selected row or column.

2. You can specify the global control by clicking on the appropriate checkboxes.

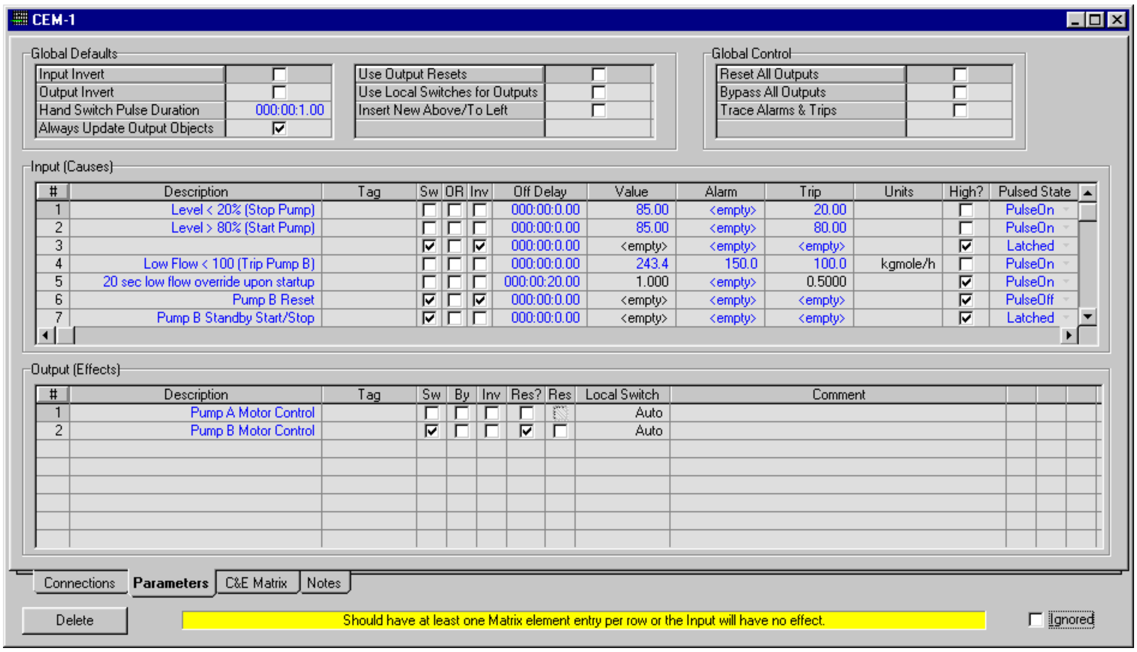
Figure 5.33



Checkbox	Description
Reset All Outputs	If you want to reset all individual outputs, select this checkbox.
Bypass All Outputs	When you select this checkbox, the **Bypass** checkbox of each output is turned on.
Trace Alarms & Trips	If you want to trace the occurrence of input alarms, input trips and output trips, select this checkbox. The Trace Alarms & Trips checkbox affects ALL Cause and Effect matrix instances in your model.

You can view all the input and output configuration information on the Parameters tab.

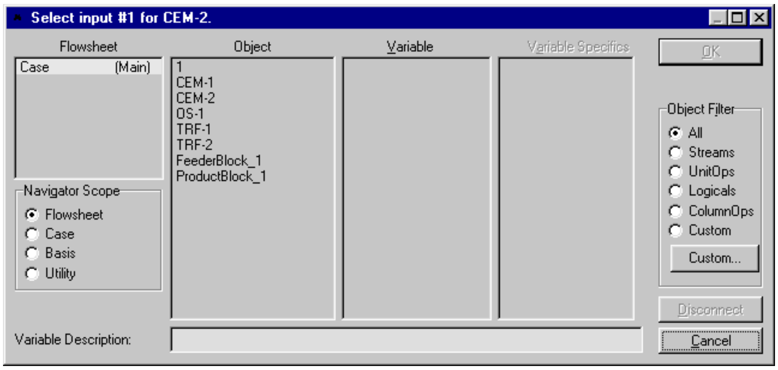
Figure 5.34



Connecting the Inputs

1. On the **Connections** tab or the **C&E Matrix** tab, click the **Add Input** button to add and connect an input. The Simulation Navigator appears.

Figure 5.35



2. From the Simulation Navigator, select the input variable. Then click **OK**.

   By default, the new input is added at the bottom of any existing inputs.

   From the Parameters tab, select the **Insert New Above/To Left** checkbox, now the new input is added above the currently selected row. You have to select the bottom blank input row to add to the bottom of the existing inputs.

3. Alternatively, If you want to add switch inputs, click the **Add Switch** button on the **C&E matrix** tab. A new input row is added, but no simulation variable needs to be selected. The user can manually change the switch during the operation of the dynamic model.

   A switch input is useful for a R(eset) matrix element entry. You should make this a Pulse On type switch. Switches can also be useful as an emergency shutdown pushbutton if you want to test your dynamic model response to the trip of a collection of outputs.

   You can change the state of the switch by clicking on the appropriate radio buttons.

Figure 5.36

For more information on viewing the specifications for the inputs and outputs, refer to the section on **Viewing the Inputs and Outputs Specifications**.

In the Inputs (Causes) table, click on a row with the SW checkbox selected.

4. Enter the description, tag, and comment (if any) for the Inputs (Causes). The description appears to the left of each input row, and its associated matrix elements on the **C&E Matrix** tab.
5. For all inputs with a simulation variable (not a switch), except for the Digital Point's OP State or an output result from a Cause and Effect matrix, specify the trip threshold. Select the **High?** checkbox if a value higher than the threshold will result in a tripped input. Otherwise, a low threshold trip is assumed.

The input can also be a time delayed Trip resulting to zero by entering a non-zero time in the Off Delay field.

You can also specify an Alarm threshold, which acts as a pre-alarm prior to the trip actually occurring.

Click the **Invert** checkbox, if you want to invert the meaning of the matrix elements.

The inversion (1 to 0 or 0 to 1) occurs at the completion of the normal input processing just before the input result is passed on for matrix processing. Hence the input status, trace messages, and so forth, occur as normal irregardless of inversion.

6. You can also override the effect of any tripped inputs by clicking on the **OR** (Override) checkbox in the Inputs (Causes) table on the **C&E Matrix** tab or the Inputs (Causes) group on the **Parameters** tab. You can use this as a startup override.

If a trip requires a reset, you will have to activate the input(s), which resets it or you may have to reset the local reset.

Connecting the Outputs

1. In the Outputs (Effects) group, click the **Add Output** button to create a new Cause and Effect Matrix column. Specify the simulation variable that you want the resultant 1 or 0 exported to.

   By default, the new output is added to the right (end) of any existing outputs. From the **Parameters** tab, select the **Insert New Above/To Left** checkbox, and now the new output is added to the left of the currently selected column. You can select the last blank output column to add to the right of the existing outputs.

You can add a new column without connecting a simulation variable, if you want to just display a trip.

You can also access the outputs result using the input in other logical operations including other Cause and Effect Matrices or using a Spreadsheet Import. Use the Simulation Navigator from that unit operation to select the Cause and Effect Matrix Output Result.

2. If you want to add a new column without connecting an object's variable, click the **Add Effect** button in the Outputs (Effects) group.
3. Select the **Reset?** checkbox if you want to specify the output has its own local reset switch. This could perhaps represent a solenoid on a shutdown valve in the field. Once the **Reset?** checkbox is selected, then the **Reset** checkbox becomes active and relevant.

It is not recommended to configure an output without a reset. This can be a matrix element R(eset) or a local reset.

4. You can also specify the presence of a local or field switch for the controlled equipment that the output is associated with. To specify the presence, select the **SW** (Switch) checkbox. Click on the appropriate radio button to set the local switch state.

Figure 5.37

> **This switch has as its permissive any inputs with a P matrix element. Also, the switch is interlocked with any inputs affecting a trip of this output.**
>
> **An output must be reset before the local switch state can be changed from off.**

5. You can click the **Invert** checkbox in the Outputs (Effects) group if the object being controlled expects a 1 to shutdown rather than a zero. This output inversion is done at the completion of the output processing, therefore the Outputs (Effects) group status bar and property view status window will show a Tripped status when sending a 1. If the trace option is turned on, a Tripped message will be traced, but a final value of 1 will be sent out.

> **When a certain input trips, causing a resulting trip in an output, there is also likely to be a cascading set of trips including other inputs which may appear to cause a trip of the original output. To detect what was the first input to cause the output's trip, you will see the relevant first out matrix element which caused the trip turn red. This colour only returns to the default of blue when the output trip has cleared and been reset.**

6. Once you have your Cause and Effect Matrix configured, you may want to use the **Bypass** checkbox of some or all outputs. This then makes the resultant value in the Outputs (Effects) group at the bottom of the **C&E Matrix** tab turn blue. This value should initialize to 1 and will remain at this until the bypass is released and matrix output processing proceeds. You can bumplessly prevent initialization trips in this manner.

Changing the order of the inputs or outputs

If the inputs or outputs are not in the order that you want, you can re-sort either the rows or columns of the Cause and Effect Matrix:

1. From the Inputs (Causes) or Outputs (Effects) tables of the **C&E Matrix tab**, select the row or column you want to move.
2. In the **Inputs (Causes)** or **Outputs (Effects)** groups, click on the row or column number displayed in the **#** field.

The row or column number is displayed in blue indicating that the user can change the value.

3. Type the new number that you want the row or column to be located at.

   If you type a number that is smaller than the number of the row (or column) you are moving, all rows (or columns) below the new number will be moved down (to the right) hence filling in the empty row (or column). Alternatively, if the new number is greater than the number of the row (or column) you are moving, the rows (or columns) will be moved up (to the left).

Viewing the Inputs and Outputs Specifications

You can view all the specifications for the inputs and outputs on the C&E Matrix tab.

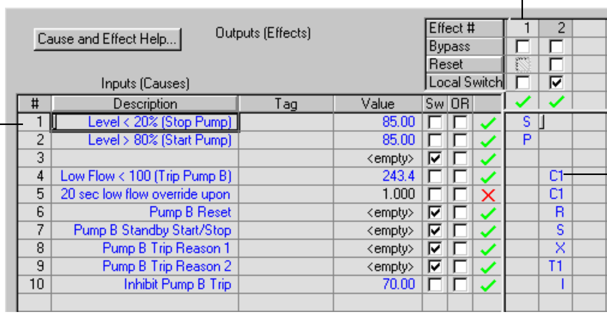
Figure 5.38

If you want to view the specifications for the Input:

1. On the C&E Matrix tab, select the row of the Cause data in the Inputs table.
2. The information for that input is shown in the Inputs group at the bottom of the tab.

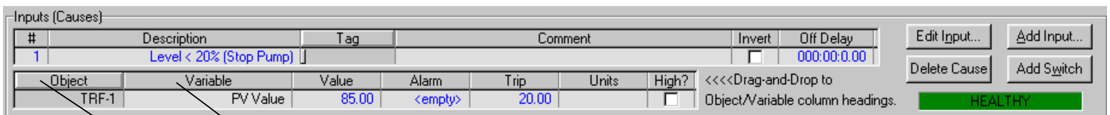
If you want to view the specifications for the Output:

1. On the C&E Matrix tab, select the column of the Effect data in the Outputs table.
2. The information for that output is shown in the Outputs group at the bottom of the tab.

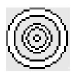


When you click on the matrix element, the specifications for the selected row and column are shown in the Inputs and Outputs groups at the bottom of the tab.

Inputs (Causes) group

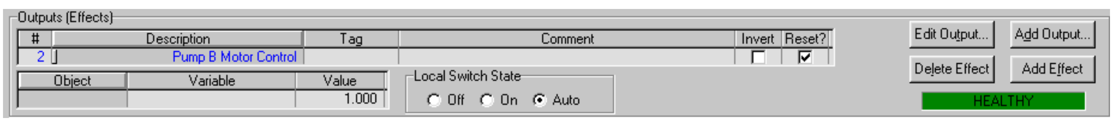


You can drag and drop input or output object/variables to the object or variable columns of the selected row or column in the Inputs (Causes) or Outputs (Effects) groups. Select the variable from some other unit operation, and then use the right mouse button to drag the selection to the desired location.

When you drag the variable the pointer changes to this cursor  .

This functionality is similar to dragging the variable in the Spreadsheet, refer to **Section 5.10 - Spreadsheet**.

Outputs (Effects) group



The C&E Matrix tab also shows the state of each input and output.

State	Description
	Healthy (1 result)
	Tripped (0 result)
	For an input this means alarm. For an output this indicates some other state. Refer to the Output status at the bottom of the property view for an indication of the exact status. For both inputs and outputs, this indicates that attention is most likely required.

Viewing the Status Messages

While integrating, the status window and the Cause and Effect Matrix's status bar may update to show the following three states:

1. one or more outputs have tripped
2. one or more inputs are in alarm
3. one or more outputs require reset (either via an input with an R matrix element or via a local output reset).

The status of the inputs and outputs is shown in the table below:

State	Inputs	Outputs
Healthy	1	1
Alarm	2	
Tripped	0	0
Reset		2
LocalReset		3
ManualOff		4
AutoOff		5

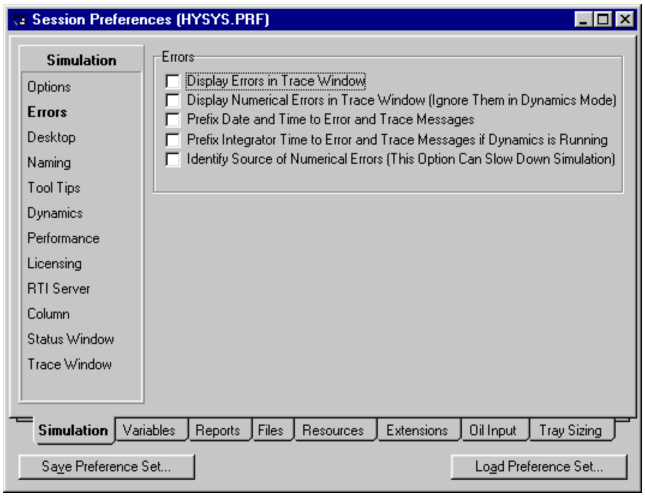
Viewing Trace Messages

You can also add a time stamp to the trace messages.

The Cause and Effect Matrix tracing is turned on by selecting the Trace & Alarms checkbox on the Global group of the Parameters tab.

1. From the **Tools** menu, select **Preferences**. The Session Preferences property view appears.
2. On the **Simulation** tab, click on the **Errors** page.

Figure 5.39



3. Click on the **Prefix Integrator Time to Error ad Trace Messages if Dynamics is Running** checkbox to add the time stamp to the trace messages, and close the Session Preferences property view.

5.4 Control Ops

HYSYS has four Control operations:

- Split Range Controller
- Ratio Controller
- PID Controller
- MPC Controller
- DMCplus Controller

5.4.1 Adding Control Operations

There are two ways that you can add Control Operations to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.

2. Click the **Logicals** radio button.
3. From the list of available unit operations, select the Control operation you want.
4. Click the **Add** button.

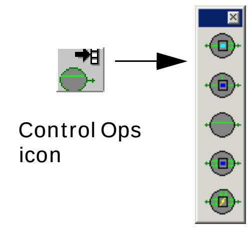
OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

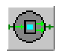
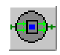
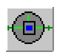
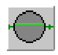
   You can also open the Object Palette by pressing **F4**.

2. Click on the **Control Ops** icon. The Controller Palette appears.

Figure 5.40

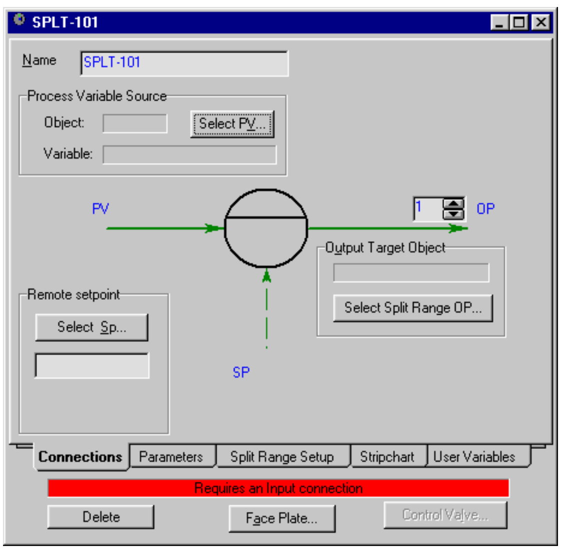


3. Double-click the icon of the Control operation that you want.

Control Operation	Icon	Control Operation	Icon
Split Range Controller		**MPC Controller**	
Ratio Controller		**DMCplus Controller**	
PID Controller			

The selected Control operations property view appears.

Figure 5.41



All Control operations contain the following buttons at the bottom of the property view:

- **Delete**. You can remove the Control ops by clicking this button.
- **Face Plate**. You can access the Face Plate property view by clicking this button.
- **Control Valve**. You can access the Control Valve property view by clicking this button.

or

- **Control OP Port**. You can access the Control OP Port property view by clicking this button.

For more information refer to **Section 5.13.2 - Controller Face Plate**.

For more information refer to **Section 5.4.7 - Control Valve**.

For more information refer to **Section 5.4.8 - Control OP Port**.

5.4.2 Split Range Controller

In the Split Range Controller, several manipulated variables are used to control a single process variable. Here both manipulated variables are driven by the output of a single controller. However, the range of operation for the manipulated variables can be independent of each other. Typical examples include the control of the pressure in a chemical reactor by manipulating the inflow and outflow from the reactor.

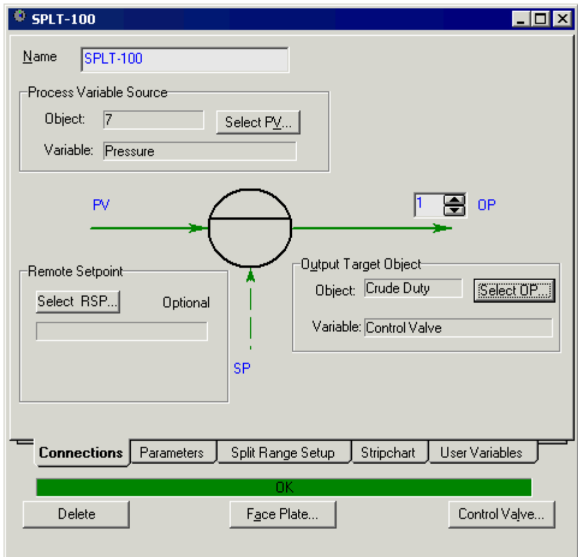
Another classic example is the temperature control of a vessel by manipulating both the cooling water flow and steam flow to the vessel.

When there is more than one controller in the strategy, for example, one single process variable with two controllers and two manipulated variables, the control is referred to as a multiple controller strategy.

In the present implementation in HYSYS there are two outputs that you have to choose. The outputs can be configured as having negative or positive gains with ranges that are independent of each other. In other words, there can be an overlap of the ranges.

The figure below shows the Split Range Controller property view.

Figure 5.42



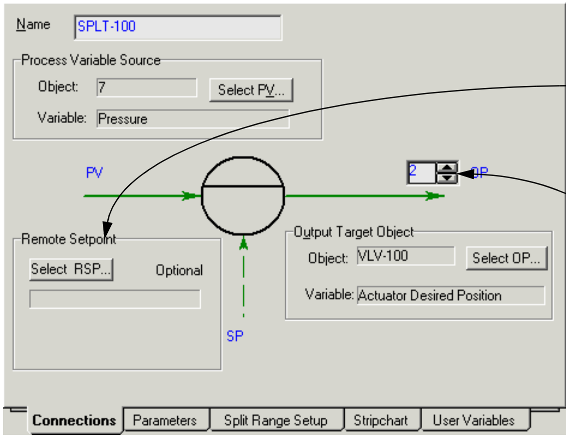
The Split Range Controller property view contains the following tabs:

- Connections
- Parameters
- Split Range Setup
- Stripchart
- User Variables

Connections Tab

On the Connections tab, you can select the process variable source and the output target objects.

Figure 5.43



Refer to **Section 1.3.9 - Variable Navigator Property View** for information Select Input PV and Select OP Object property view.

Object	Description
Name field	Allows you to change the name of the operation.
Process Variable Source group	• **Select PV** button enables you to access the Select Input PV property view and select the source object of the Process Variable. • **Object** field displays the Process Variable object (stream or operation) that owns the variable you want to compare. • **Variable** field displays the variable of the selected object.
Output Target Object group	• **Select OP** button enables you to access the Select OP Object property view and select the source object of the Output Target. • **Object** field displays the object (stream or operation) that is controlled by the operation. • **Variable** field displays the variable of the selected object.
Remote Setpoint group	• **Select RSP** button enables you to access the Select Remote Setpoint property view and select the source object of the Remote Setpoint. • **Remote Setpoint** field displays the selected master controller.

Parameters Tab

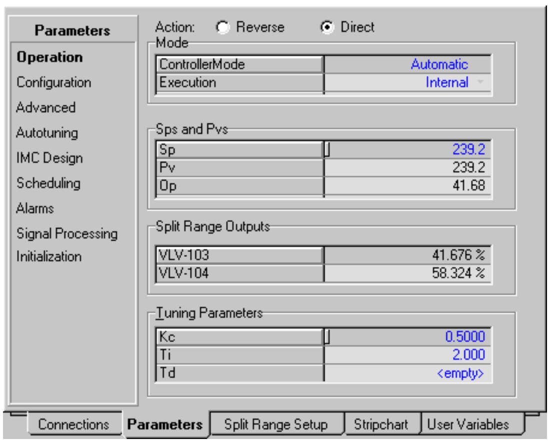
The Parameters tab contains the following pages:

- Operation
- Configuration
- Advanced
- Autotuning
- IMC Design
- Scheduling
- Alarms
- Signal Processing
- Initialization

Operation Page

On the Operation page, you can manipulate how the operation reacts to the process variable inputs.

Figure 5.44



Object	Description
Action	You can select one of the two types of action available for the operation to take when the process variable value deviates from the setpoint value: • **Direct**. When the PV rises above the SP, the OP increases. When the PV falls below the SP, the OP decreases. • **Reverse**. When the PV rises above the SP, the OP decreases. When the PV falls below the SP, the OP increases.
Controller Mode	You can select from three types of controller mode: • **Off**. The operation does not manipulate the control valve, although the appropriate information is still tracked. • **Manual**. Manipulate the operation output manually. • **Automatic**. The operation reacts to fluctuations in the Process Variable and manipulates the Output according to the logic defined by the tuning parameters.
Execution	You can select from two types of execution. • **Internal.** Confines the signals generated to stay within HYSYS. • **External.** Sends the signals to a DCS, if a DCS is connected to HYSYS.
Sp	Allows you to specify the setpoint value.
Pv	Displays the process variable value.
Op	Displays the output value.
Split Range Output	Displays the current OP value in percent for each output.
Kc (Gain)	Allows you to specify the proportional gain of the operation.
Ti (Reset)	Allows you to specify the integral (reset) time of the operation.
Td (Derivative)	Allows you to specify the derivative (rate) time of the operation.

Refer to **Tuning Parameters Group** section for more information on Kc, Ti, and Td.

Tuning Parameters Group

The Tuning Parameters group allows you to define the constants associated with the PID control equation. The characteristic equation for a PID Controller is given below:

$$OP(t) = OP_{ss} + K_c E(t) + \frac{K_c}{T_i}\int E(t)\mathrm{dt} + K_c T_d \frac{dE(t)}{dt} \qquad (5.4)$$

where:

$OP(t)$ = *controller output at time t*

OP_{ss} = *steady state controller output (at zero error)*

$E(t)$ = *error at time t*

K_c = *proportional gain of the controller*

T_i = *integral (reset) time of the controller*

T_d = *derivative (rate) time of the controller*

The error at any time is the difference between the Setpoint and the Process Variable:

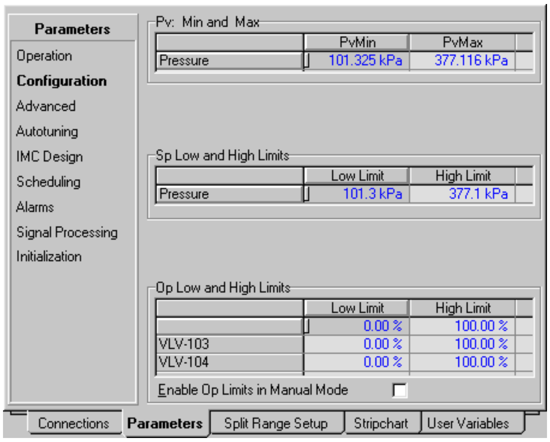
$$E(t) = SP(t) - PV(t) \qquad (5.5)$$

Depending on which of the three tuning parameters you have specified, the Controller responds accordingly to the error. A Proportional-only controller is modeled by providing only a value for K_p, while a PI (Proportional-Integral) Controller requires values for K_p and T_i. Finally, the PID (Proportional-Integral-Derivative) Controller requires values for all three of K_p, T_i, and T_d.

Configuration Page

The Configuration page allows you to specify the process variable, setpoint, and output ranges.

Figure 5.45



PV: Min and Max Group

For the operation to become operational, you must:

1. Define the minimum and maximum values for the PV (the operation cannot switch from Off mode unless PVmin and PVmax are defined).
2. Once you provide these values (as well as the Control Valve span), you can select the Automatic mode and give a value for the Setpoint.

HYSYS uses the current value of the PV as the set point by default, but you can change this value at any time.

Without a PV span, the Controller cannot function.

HYSYS converts the PV range into a 0-100% range, which is then used in the solution algorithm. The following equation is used to translate a PV value into a percentage of the range:

$$PV(\%) = \left(\frac{PV - PV_{min}}{PV_{max} - PV_{min}}\right)100 \tag{5.6}$$

SP Low and High Limits Group

In this group, you can specify the higher and lower limits for the setpoints to reflect your needs and safety requirements. The setpoint limits enforce an acceptable range of values that could be entered via the interface or from a remote source. By default the PVs min. and max values are used as the SPs low and high limits, respectively.

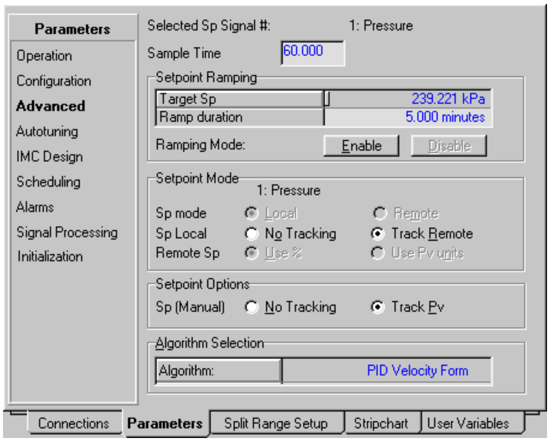
Op Low and High Limits Group

In this group, you can specify the higher and lower limits for all the outputs. The output limits ensure that a predetermined minimum, or maximum output value is never exceeded. By default 0% and 100% is selected as a low and a high of limit, respectively for all the outputs.

When the Enable Op Limits in Manual Mode checkbox is selected, you can enable the set point and output limits when in manual mode.

Advanced Page

Figure 5.46



The Advanced page contains the following four groups described in the table below:

Group	Description
Setpoint Ramping	Allows you to specify the ramp target and duration.
Setpoint Mode	Contains the options for setpoint mode and tracking, as well as, the option for remote setpoint.
Setpoint Options	Contains the option for setpoint tracking only in manual mode.
Algorithm Selection	Contains the PID controller algorithms for output calculation.

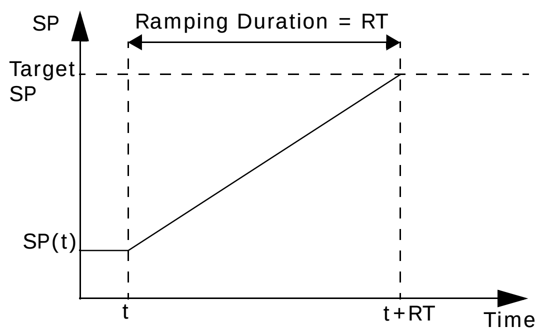
The setpoint signal is specified in the **Selected Sp Signal #** field by clicking the up or down arrow button, or by typing the appropriate number in the field.

Depending upon the signal selected, the page displays the respective setpoint settings.

Setpoint Ramping Group

The setpoint ramping function has been modified in the present MPC controllers. Now it is continuous, in other words, when set to on by clicking the **Enable** button, the setpoint changes over the specified period of time in a linear manner.

Figure 5.47



Controller Ramping

> **Setpoint ramping is only available in Auto mode.**

The Setpoint Ramping group contains the following two fields:

- **Target SP**. Contains the Setpoint you want the Controller to have at the end of the ramping interval. When the ramping is turn off, the Target SP field display the same value as SP field on the Configuration page.
- **Ramping**. Contains the time interval you want to complete setpoint change in.

Besides these two fields there are also two buttons available in this group:

- **Enable**. Activates the ramping process.
- **Disable**. Stops the ramping process.

While the controller is in ramping mode, you can change the setpoint as follows:

- Enter a new setpoint in the Target SP field
- Enter a new setpoint in the SP field, on the Operation page.

During the setpoint ramping the Target SP field shows the final value of the setpoint whereas the SP field, on the Operation page, shows the current setpoint seen internally by the control algorithm.

During ramping, if a second setpoint change has been activated, then Ramping Duration time would be restarted for the new setpoint.

Setpoint Mode Group

You have now the ability to switch the setpoint from local to remote using the Setpoint mode radio buttons. Essentially, there are two internal setpoints in the controller, the first is the local setpoint where the you can manually specify the setpoint via the property view (interface), and the other is the remote setpoint which comes from another object such as a spreadsheet or another controller cascading down a setpoint, in other words, a master in the classical cascade control scheme.

The Sp Local option allows you to disable the tracking for the local setpoint when the controller is placed in manual mode. You can also have the local setpoint track the remote setpoint by selecting the Track Remote radio button.

The Remote Sp option allows you to select either the **Use%** radio button (for restricting the setpoint changes to be in percentage) or **Use Pv units** radio button (for setpoint changes to be in Pv units).

- If the Remote Sp is set to **Use%**, then the controller reads in a value in percentage from a remote source, and using the Pv range calculates the new setpoint.
- If the Remote Sp is set to **Use Pv units**, then the controller reads in a value from a remote source, and sets a new setpoint. The remote source's setpoint must have the same units as the controller Pv.

SetPoint Options Group

If you select the Track PV radio button then there is automatic setpoint tracking in manual mode, that sets the value of the setpoint equal to the value of the Pv prior to the controller being placed in the manual mode. This means that upon switching from manual to automatic mode the values of the setpoint and Pv were equal and, therefore, there was an automatic bumpless transfer.

Also you have the option not to track the Pv, by selecting the No Tracking radio button, when the controller is placed in manual mode. However, when the controller is switched into the automatic mode from manual, there is an internal resetting of the controller errors to ensure that there is an instantaneous bumpless transfer prior to the controller recognizing a setpoint that is different from the Pv.

Algorithm Selection Group

In the Algorithm Selection group you can select one of the three available controller update algorithms:

- PID Velocity Form
- PID Positional Form (ARW = Anti-Reset Windup)
- PID Positional Form (noARW)

Velocity or Differential Form

The velocity or differential form of the controller should be applied when there is an integral term. When there is no integral term a positional form of the controller should be used.

In the velocity or differential form the controller equation is given as:

$$u(t) = u(t-1) + K_c\left[e(t) - e(t) + \frac{1}{T_i}e(kh) + T_d\frac{(e(t) - 2e(t-1) + e(t-2))}{h}\right] \qquad (5.7)$$

where:

u(t) = controller output and t is the enumerated sampling instance in time

u(t-1) = value of the output one sampling period ago

Kc, Ti, and Td = controller parameters

h = sampling period

Positional Form

In the positional form of the algorithm, the controller output is given by:

$$u(t) = K_c\left[e(t) - e(t) + \frac{1}{T_i}\sum_{k=0}^{n} e(kh) + T_d\frac{(e(t) - e(t-1))}{h}\right] \qquad (5.8)$$

Here it is important to handle properly the summation term associated with the integral part of the control algorithm. Specifically, the integral term could grow to a very large value in instances where the output device is saturated, and the PV is still not able to get to the setpoint. For situations like the one above, it is important to reset the value of the summation to ensure that the output is equal to the limit (upper or lower) of the controller output. As such, when the setpoint is changed to a region where the controller can effectively control, the controller responds immediately without having to decrease a summation term that has grown way beyond the upper or lower limit of the output. This is referred to as an automatic resetting of the control integral term commonly called anti-reset windup.

In HYSYS both algorithms are implemented as presented above with one key exception, there is no derivative kick. This means that the derivative part of the control algorithm operates on the process variable as opposed to the error term.

As such the control equation given in **Equation (5.7)** is implemented as follows:

$$u(t) = u(t-1) + K_c\left[e(t) - e(t) + \frac{1}{T_i}e(kh) + T_d\frac{(-pv + 2pv(t-1) - pv(t-2))}{h}\right] \qquad (5.9)$$

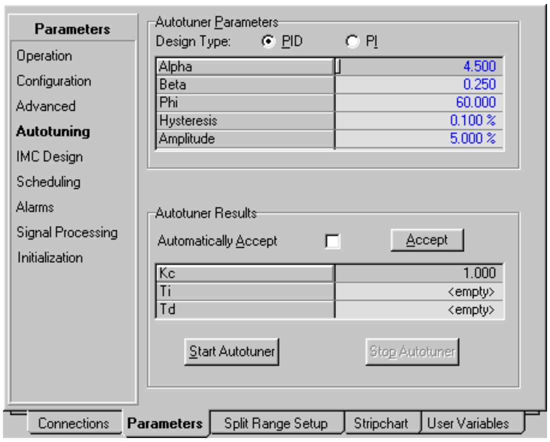
Autotuning Page

For more information about autotuning parameters, refer to the **Autotuner Page** in **Section 5.4.4 - PID Controller**.

You can set the autotuning parameters on the Autotuning page. This page consists two groups:

- **Autotuner Parameters**. Contains the parameters required by the Autotuner to calculate the controller parameters.
- **Autotuner Results**. Displays the resulting controller parameters. You have the option to accept the results as the current tuning parameters.

Figure 5.48



Autotuner Parameters Group

> **In the present version of the software there are default values specified for the PID tuning. Before starting the autotuner, you must ensure that the controller is in the manual or automatic mode, and the process is relatively steady.**
>
> **If you move the cursor over the tuning parameters field, the Status Bar displays the parameters range.**

In this group, you can specify the controller type by selecting the PID radio button or the PI radio button for the Design Type.

In the present autotuner implementation there are five parameters that you must specify which are as follows:

Parameter	Range
Ratio (Ti/Td) (Alpha)	$3.0 \le \alpha \le 6.0$
Gain ratio (Beta)	$0.10 \le \beta \le 1.0$
Phase angle (Phi)	$30° \le \phi \le 65°$
Relay hysteresis (h)	$0.01\% \le h \le 5.0\%$
Relay amplitude (d)	$0.5\% \le d \le 10.0\%$

Autotuner Results Group

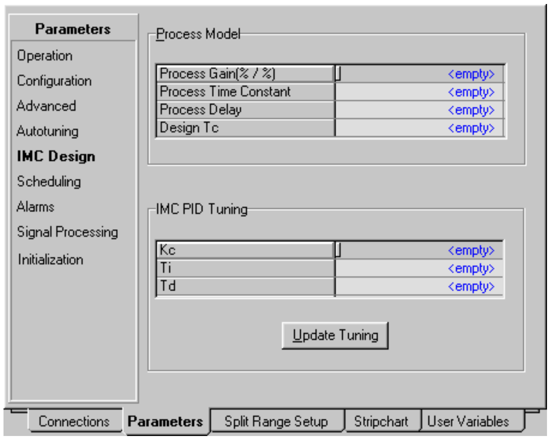
This group displays the results of the autotuner calculation, and allows you to accept the results as the current controller setting. The **Start Autotuner** button activates the tuning calculation, and the **Stop Autotuning** button aborts the calculations.

After running the autotuner, you have the option to accepts the results either automatically or manually. Selecting the **Automatically Accept** checkbox sets the resulting controller parameters as the current value instantly. If the Automatically Accept checkbox is inactive, you can specify the calculated controller parameters to be the current setting by clicking the Accept button.

IMC Design Page

The IMC Design page allows you to use the internal model control (IMC) calculator to calculate the operation parameters based on a specified model of the process one is attempting to control.

Figure 5.49



The IMC method is quite common in most of the process industries and has a very solid theoretical basis. In general, the performance obtained using this design methodology is superior to most of the existing techniques for tuning PIDs. As such, when there is a process model available (first order plus delay) this approach should be used to determine the controller parameters. You must specify a design time constant, which is usually chosen as three times that of the measured process time constant.

The IMC Design page has the following two groups:

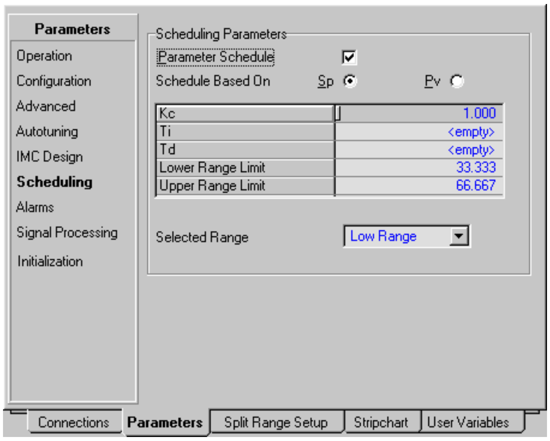
Group	Description
Process Model	Contains the parameters for the process model, which are required by the IMC calculator. • Process Gain • Process Time Constraint • Process Delay • Design To
IMC PID Tuning	Displays the operation parameters.

As soon as you enter the parameters in the Process Model group, the operation parameters are calculated and displayed in the IMC PID Tuning group. You can accept them as the current tuning parameters by clicking the Update Tuning button.

Scheduling Page

The Scheduling page gives you the ability to do parameter scheduling. This feature is quite useful for nonlinear processes where the process model changes significantly over the region of operation.

Figure 5.50



The parameter scheduling is activated through the **Parameter Schedule** checkbox. You can use three different sets of PID parameters, if you so desires for three different regions of operation.

The following regions of operation can be specified from the Selected Range drop-down list.

- Low Range
- Middle Range
- High Range

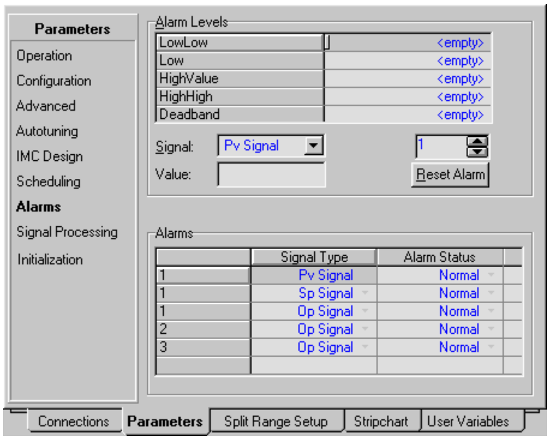
These regions of operations can be based either on the setpoint, or PV of the controller. The ranges can also be specified, the default values are 0-33%, 33%-66%, and 66%-100% of the selected scheduling signal. You need to specify the middle range limit by defining the Upper and Lower Range Limits.

The values of 0 and 100 cannot be specified for both the Lower and the Upper Range Limits.

Alarms Page

The Alarms page allows you to set alarm limits on all exogenous inputs to and outputs from the controller.

Figure 5.51



The Alarms page contains two groups:

- Alarm Levels
- Alarms

Alarm Levels Group

The Alarm Level group allows you to set, and configure the alarm points for a selected signal type. There are four alarm points that could be configured:

- LowLow
- Low
- High
- HighHigh

The alarm points should be specified in the descending order from HighHigh to LowLow points. You cannot specify the value of the Low and LowLow alarm points to be higher than the signal value. Similarly, the High and HighHigh alarm points cannot be specified to a value lower than the signal value. Also, no two alarm points can be specified to a similar value. In addition, you can specify a deadband for a given set of alarms. This can be helpful in situations where the signal is “noisy” to avoid constant triggering of the alarm. If a deadband is specified, you have to specify the alarm points so that their difference is greater than the deadband. At present the range for the allowable deadband is as follows:

$0.0\% \leq \text{deadband} \leq 1.5\%$ of the signal range.

> **The above limits are set internally, and are not available for adjustment by the user.**

Alarms Group

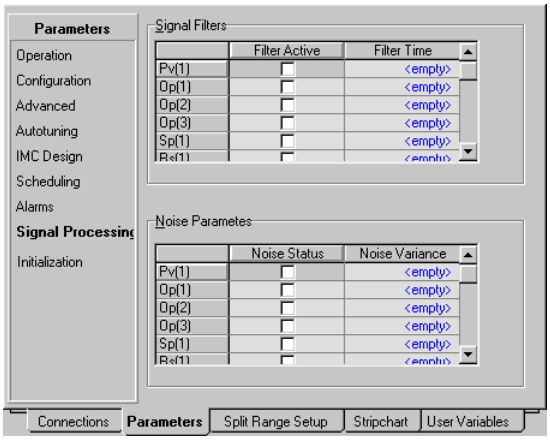
The Alarms group displays the recently violated alarm for the following signals:

Signal	Description
PV	Process Variable
OP	Output
SP	Setpoint

Signal Processing Page

The Signal Processing page allows you to add filters to any signal associated with the operation, as well as test the robustness of any tuning on the controller.

Figure 5.52



This page consists of two groups:

- Signal Filters
- Noise Parameters

Both of these groups allow you to filter, and test the robustness of the following tuning parameters:

- Pv
- Op
- Sp
- Dv
- Rs

To apply the filter select the checkbox corresponding to the signal you want to filter. Once active, you can specify the filter time. As you increase the filter time you are filtering out frequency information from the signal.

For example, the signal is noisy, there is a smoothing effect noticed on the plot of the PV. Notice that it is possible to add a filter that makes the controller unstable. In such cases the controller needs to be returned. Adding a filter has the same effect as changing the process, which the controller is trying to control.

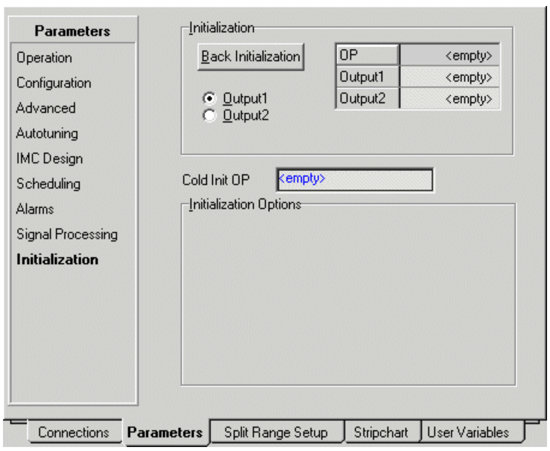
Activating a Noise Parameter is done the same way as adding a filter. However, instead of specifying a filter time you are specifying a variance. Notice that if a high variance on the PV signal is chosen the controller may become unstable. As you increase the noise level for a given signal you observe a some what random variation of the signal.

Initialization Page

The Initialization page allows you to initialize an appropriate OP value to start the controller smoothly. To back initialize the controller, click on the Back Initialization button and HYSYS will initialize the controller output based on the current position of the executor (for example, a valve or another controller). The current back initialization OP value is displayed in the OP value field.

Since the split controller has two outputs (two OP values), you can click on the Output1 or Output2 radio button to chose which OP value you want to use to back initialize the controller.

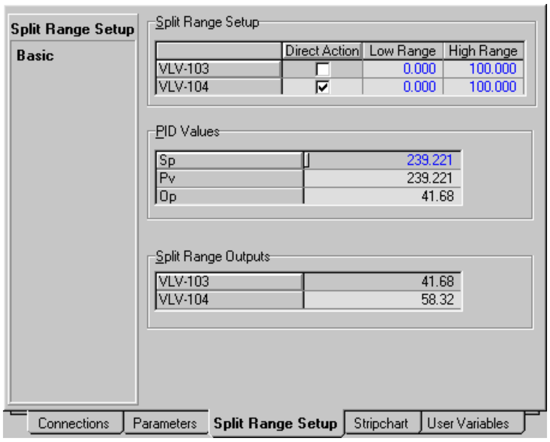
Figure 5.53



Split Range Setup Tab

The Split Range Setup tab allows you to specify the split ranges for the controller. The Split Range Setup tab consists of three groups: Split Range Setup, PID Values, and Split Range Outputs.

Figure 5.54



Stripchart Tab

Refer to **Section 1.3.7 - Stripchart Page / Tab** for more information.

The Stripchart tab allows you to select and create default strip charts containing various variable associated to the operation.

User Variables Tab

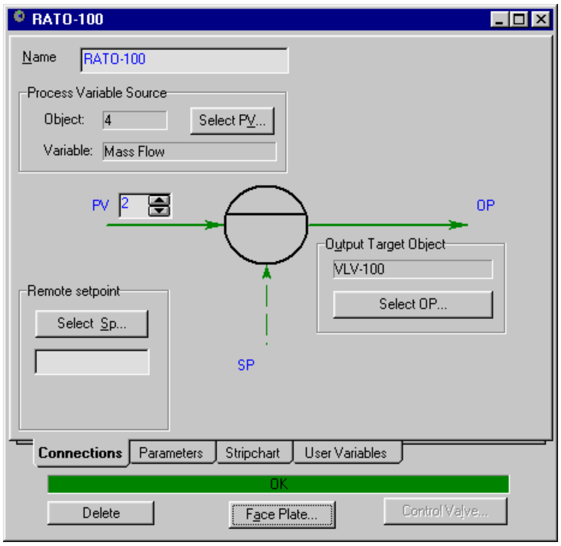
For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables tab enables you to create and implement your own user variables for the current operation.

5.4.3 Ratio Controller

In the Ratio Controller the objective is to keep the ratio of two variables, the load and the manipulated, constant.

Figure 5.55



The Ratio Controller is a special type of feedforward control, and can be implemented in two ways:

- **Method 1**. The actual ratio of the two variables is calculated using a divider, and is sent on to the ratio controller in which the setpoint is the required ratio.
- **Method 2**. The value of the load variable is measured and sent to a ratio station, which then calculates the setpoint of the manipulated (second) variable.

The inclusion of a divider in approach (method 1) renders the methodology less desirable since it results in a loop in which the process gain varies in a nonlinear manner as a result of the included divider. As such, method (2) is the preferred way of doing the ratio implementation, and is the approached followed in this implementation for HYSYS.

The Ratio Controller property view contains the following tabs:

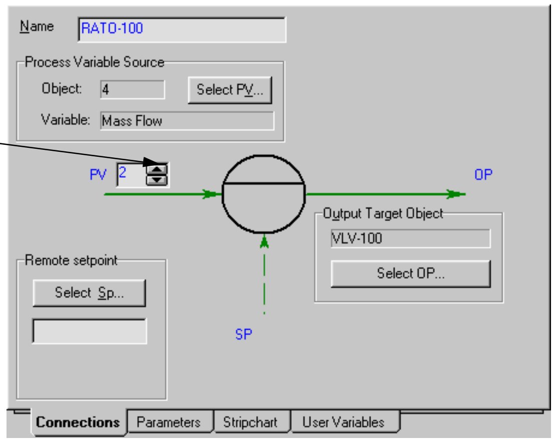
- Connections
- Parameters
- Stripchart
- User Variables

Connections Tab

On the Connections tab, you can select the process variable source, and the output target object. You can also select a remote setpoint value.

Figure 5.56

Click the up and down arrows to access the first and second variables.



Object	Description
Name	Allows you to change the name of the operation.
Process Variable Source: Object	Contains the Process Variable Object (stream or operation) that owns the variable you want to compare.
Process Variable Source: Variable	Contains the Process Variable you want to compare.
Output Target Object	The stream or valve, which is controlled by the operation.
Select PV/OP	These two buttons open the Variable Navigator which selects the Process Variable and the Output Target Object respectively.
Remote Setpoint Source	If you are using set point from a remote source, select the remote Setpoint Source associated with the Master controller

Refer to **Section 1.3.9 - Variable Navigator Property View** for information.

Parameters Tab

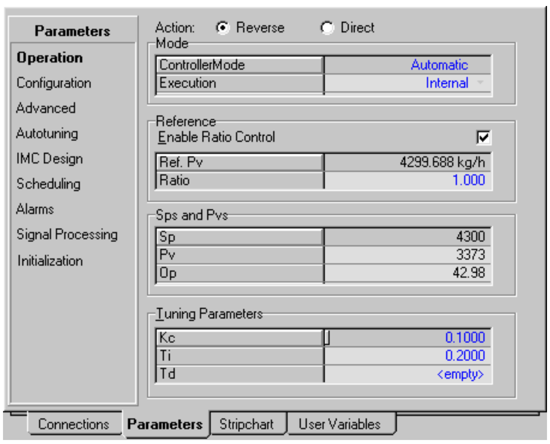
The Parameters tab contains the following pages:

- Operation
- Configuration
- Advanced
- Autotuning
- IMC Design
- Scheduling
- Alarms
- Signal Processing
- Initialization

Operation Page

On the Operation page, you can manipulate how the operation reacts to the process variable inputs.

Figure 5.57



Object	Description
Action	You can select one of the two types of action available for the operation to take when the process variable value deviates from the setpoint value: • **Direct**. When the PV rises above the SP, the OP increases. When the PV falls below the SP, the OP decreases. • **Reverse**. When the PV rises above the SP, the OP decreases. When the PV falls below the SP, the OP increases.
Controller Mode	You can select from three types of controller mode: • **Off**. The operation does not manipulate the control valve, although the appropriate information is still tracked. • **Manual**. Manipulate the operation output manually. • **Automatic**. The operation reacts to fluctuations in the Process Variable and manipulates the Output according to the logic defined by the tuning parameters.
Execution	You can select from two types of execution. • **Internal.** Confines the signals generated to stay within HYSYS. • **External.** Sends the signals to a DCS, if a DCS is connected to HYSYS.
Enable Ratio Control	This checkbox has to be selected, if you want to set the ratio value for the operation. If this checkbox is inactive, HYSYS calculates the ratio value between the two selected process variables.
Ref. Pv	The value in this field is used to calculate the setpoint along with the ratio.
Ratio	Displays the set or calculated ratio value between the selected the two process variables.
Sp	Allows you to specify the setpoint value.
Pv	Displays the process variable value.
Op	Displays the output value.
Gain	Allows you to specify the proportional gain of the operation.
Reset	Allows you to specify the integral (reset) time of the operation.
Derivative	Allows you to specify the derivative (rate) time of the operation.

Refer to the **Tuning Parameters Group** section for more information on Gain, Reset, and Derivative.

Tuning Parameters Group

The Tuning Parameters group allows you to define the constants associated with the PID control equation. The characteristic equation for a PID Controller is given below:

$$OP(t) = OP_{ss} + K_c E(t) + \frac{K_c}{T_i}\int E(t)\mathrm{dt} + K_c T_d \frac{dE(t)}{dt} \qquad (5.10)$$

where:

$OP(t)$ = *controller output at time t*

OP_{ss} = *steady state controller output (at zero error)*

$E(t)$ = *error at time t*

K_c = *proportional gain of the controller*

T_i = *integral (reset) time of the controller*

T_d = *derivative (rate) time of the controller*

The error at any time is the difference between the Setpoint and the Process Variable:

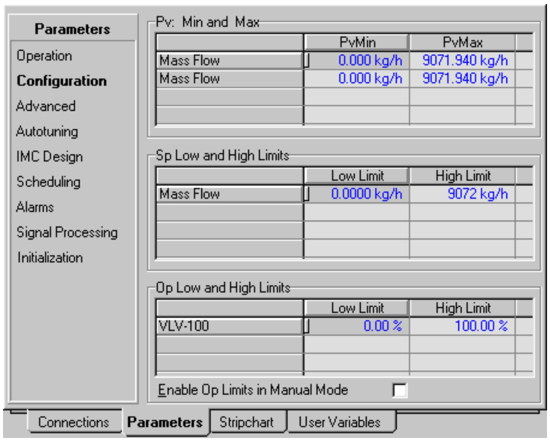
$$E(t) = SP(t) - PV(t) \qquad (5.11)$$

Depending on which of the three tuning parameters you have specified, the Controller responds accordingly to the error. A Proportional-only controller is modeled by providing only a value for K_p, while a PI (Proportional-Integral) Controller requires values for K_p and T_i. Finally, the PID (Proportional-Integral-Derivative) Controller requires values for all three of K_p, T_i, and T_d.

Configuration Page

The Configuration page allows you to specify the process variable, setpoint, and output ranges.

Figure 5.58



PV: Min and Max Group

For the operation to become operational, you must:

1. Define the minimum and maximum values for the PV (the operation cannot switch from Off mode unless PVmin and PVmax are defined).
2. Once you provide these values (as well as the Control Valve span), you can select the Automatic mode and give a value for the Setpoint.

HYSYS uses the current value of the PV as the set point by default, but you can change this value at any time.

Without a PV span, the Controller cannot function.

HYSYS converts the PV range into a 0-100% range, which is then used in the solution algorithm.

The following equation is used to translate a PV value into a percentage of the range:

$$PV(\%) = \left(\frac{PV - PV_{min}}{PV_{max} - PV_{min}}\right)100 \qquad (5.12)$$

SP Low and High Limits Group

In this group, you can specify the higher and lower limits for the Setpoints to reflect your needs and safety requirements. The Setpoint limits enforce an acceptable range of values that could be entered via the interface or from a remote source. By default the PVs min. and max values are used as the SPs low and high limits, respectively.

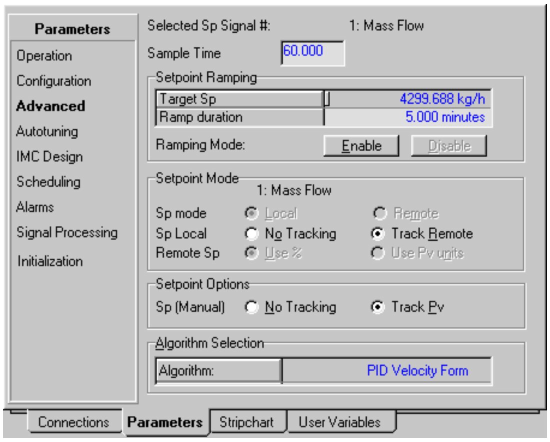
Op Low and High Limits Group

In this group, you can specify the higher and lower limits for all the outputs. The output limits ensure that a predetermined minimum or maximum output value is never exceeded. By default 0% and 100% is selected as a low and a high of limit, respectively for all the outputs.

When the **Enable Op Limits in Manual Mode** checkbox is selected, you can enable the set point and output limits when in manual mode.

Advance Page

Figure 5.59



The Advanced page contains the following four groups:

Group	Description
Setpoint Ramping	Allows you to specify the ramp target and duration.
Setpoint Mode	Contains the options for setpoint mode and tracking as well as the option for remote setpoint.
Setpoint Options	Contains the option for setpoint tracking only in manual mode.
Algorithm Selection	Contains the PID controller algorithms for output calculation.

The setpoint signal is specified in the **Selected Sp Signal #** field by clicking the up or down arrow button, or typing the appropriate number in the field.

Depending upon the signal selected, the Advance page displays the respective setpoint settings.

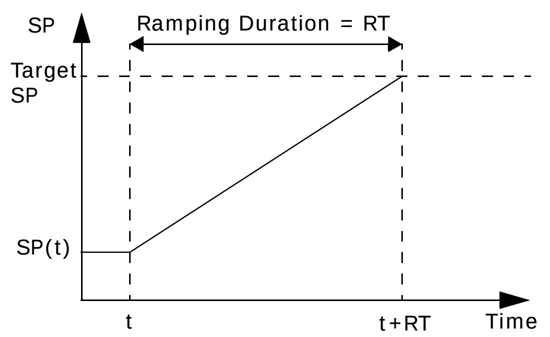
Setpoint Ramping Group

The setpoint ramping function has been modified in the present MPC controllers. Now it is continuous, in other words, when set to on by clicking the Enable button, the setpoint changes over the specified period of time in a linear manner. The Setpoint Ramping group contains the following two fields:

Field	Input Required
Target SP	Contains the Setpoint you want the Controller to have at the end of the ramping interval. When the ramping is turn off, the Target SP field display the same value as SP field on the Configuration page.
Ramping Duration	Contains the time interval you want to complete setpoint change in.

Setpoint ramping is only available in Auto mode.

Figure 5.60



Controller Ramping

Besides these two fields there are also two buttons available in this group:

- **Enable**. Activates the ramping process.
- **Disable**. Stops the ramping process.

While the controller is in ramping mode, you can change the setpoint as follows:

- Enter a new setpoint in the Target SP field.
- Enter a new setpoint in the SP field, on the Operation page.

During the setpoint ramping the Target SP field shows the final value of the setpoint whereas the SP field, on the Operation page, shows the current setpoint seen internally by the control algorithm.

During ramping, if a second setpoint change has been activated, then Ramping Duration time would be restarted for the new setpoint.

Setpoint Mode Group

You have now the ability to switch the setpoint from local to remote using the Setpoint mode radio buttons. Essentially, there are two internal setpoints in the controller, the first is the local setpoint where you can manually specify the setpoint via the property view (interface), and the other is the remote setpoint which comes from another object such as a spreadsheet or another controller cascading down a setpoint, in other words, a master in the classical cascade control scheme.

The Sp Local option allows you to disable the tracking for the local setpoint when the controller is placed in manual mode. You can also have the local setpoint track the remote setpoint by selecting the Track Remote radio button.

The Remote Sp option allows you to select either the **Use %** radio button (for restricting the setpoint changes to be in percentage) or **Use Pv units** radio button (for setpoint changes to be in Pv units).

- If the Remote Sp is set to **Use %**, then the controller reads in a value in percentage from a remote source, and using the Pv range calculates the new setpoint.
- If the Remote Sp is set to **Use Pv units**, then the controller reads in a value from a remote source and sets a new setpoint. The remote source's setpoint must have the same units as the controller Pv.

SetPoint Options Group

If you select the Track PV radio button, then there is automatic setpoint tracking in manual mode, that sets the value of the setpoint equal to the value of the Pv prior to the controller being placed in the manual mode. This means that upon switching from manual to automatic mode the values of the setpoint and

Pv were equal and, therefore, there was an automatic bumpless transfer.

Also you have the option not to track the pv, by clicking the **No Tracking** radio button, when the controller is placed in manual mode. However, when the controller is switched into the automatic mode from manual, there is an internal resetting of the controller errors to ensure that there is an instantaneous bumpless transfer prior to the controller recognizing a setpoint that is different from the Pv.

Algorithm Selection Group

In the Algorithm Selection group you can select one of three available controller update algorithms:

- PID Velocity Form
- PID Positional Form (ARW = Anti-Reset Windup)
- PID Positional Form (noARW)

Velocity or Differential Form

The velocity or differential form of the controller should be applied when there is an integral term. When there is no integral term a positional form of the controller should be used.

In the velocity or differential form the controller equation is given as:

$$u(t) = u(t-1) + K_c\left[e(t) - e(t) + \frac{1}{T_i}e(kh) + T_d\frac{(e(t) - 2e(t-1) + e(t-2))}{h}\right] \qquad (5.13)$$

where:

u(t) = controller output and t is the enumerated sampling instance in time

u(t-1) = value of the output one sampling period ago

Kc, Ti, and Td = controller parameters

h = sampling period

Positional Form

In the positional form of the algorithm, the controller output is given by:

$$u(t) = K_c\left[e(t) - e(t) + \frac{1}{T_i}\sum_{k=0}^{n} e(kh) + T_d\frac{(e(t) - e(t-1))}{h}\right] \tag{5.14}$$

Here it is important to handle properly the summation term associated with the integral part of the control algorithm. Specifically, the integral term could grow to a very large value in instances where the output device is saturated and the PV is still not able to get to the setpoint.

For situations like the one above, it is important to reset the value of the summation to ensure that the output is equal to the limit (upper or lower) of the controller output. As such, when the setpoint is changed to a region where the controller can effectively control, the controller responds immediately without having to decrease a summation term that has grown way beyond the upper or lower limit of the output. This is referred to as an automatic resetting of the control integral term commonly called anti-reset windup.

In HYSYS both algorithms are implemented as presented above with one key exception, there is no derivative kick. This means that the derivative part of the control algorithm operates on the process variable as opposed to the error term.

As such the control equation given in **Equation (5.13)** is implemented as follows:

$$u(t) = u(t-1) + K_c\left[e(t) - e(t) + \frac{1}{T_i}e(kh) + T_d\frac{(-pv + 2pv(t-1) - pv(t-2))}{h}\right] \tag{5.15}$$

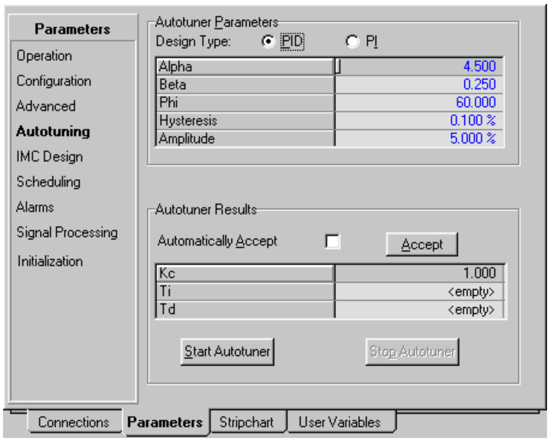
Autotuning Page

For more information about autotuning parameters, refer to the **Autotuner Page** in **Section 5.4.4 - PID Controller**.

You can set the autotuning parameters on the Autotuning page. This page consists of two groups:

- **Autotuner Parameters**. Contains the parameters required by the Autotuner to calculate the controller parameters.
- **Autotuner Results**. Displays the resulting controller parameters. You have the option to accept the results as the current tuning parameters.

Figure 5.61



Autotuner Parameters Group

In this group, you can specify the controller type by selecting the PID radio button or the PI radio button for the Design Type. In the present autotuner implementation there are five parameters that you must specify, which are as follows:

Parameter	Range
Ratio (Ti/Td) (Alpha)	$3.0 \le \alpha \le 6.0$
Gain ratio (Beta)	$0.10 \le \beta \le 1.0$
Phase angle (Phi)	$30° \le \phi \le 65°$

Parameter	Range
Relay hysteresis (h)	$0.01\% \le h \le 5.0\%$
Relay amplitude (d)	$0.5\% \le d \le 10.0\%$

In the present version of the software there are default values specified for the PID tuning. Before starting the autotuner, you must ensure that the controller is in the manual or automatic mode, and the process is relatively steady.

If you move the cursor over the tuning parameters field, the Status Bar displays the parameters range.

Autotuner Results Group

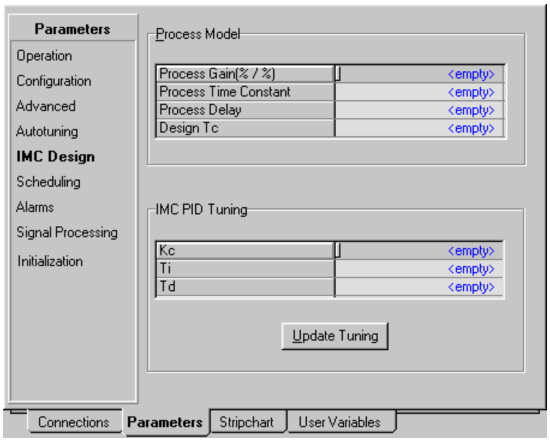
This group displays the results of the autotuner calculation and allows you to accept the results as the current controller setting. The **Start Autotuner** button activates the tuning calculation, and the **Stop Autotuning** button abort the calculations.

After running the autotuner, you have the option to accept the results either automatically or manually. Selecting the **Automatically Accept** checkbox sets the resulting controller parameters as the current value instantly. If the Automatically Accept checkbox is inactive, you can specify the calculated controller parameters to be the current setting by clicking the Accept button.

IMC Design Page

The IMC Design page allows you to use the internal model control (IMC) calculator to calculate the operation parameters based on a specified model of the process one is attempting to control.

Figure 5.62



The IMC method is quite common in most of the process industries, and has a very solid theoretical basis. In general, the performance obtained using this design methodology is superior to most of the existing techniques for tuning PIDs. As such, when there is a process model available (first order plus delay) this approach should be used to determine the controller parameters. You must specify a design time constant, which is usually chosen as three times that of the measured process time constant.

The IMC Design page has the following two groups described in the table below:

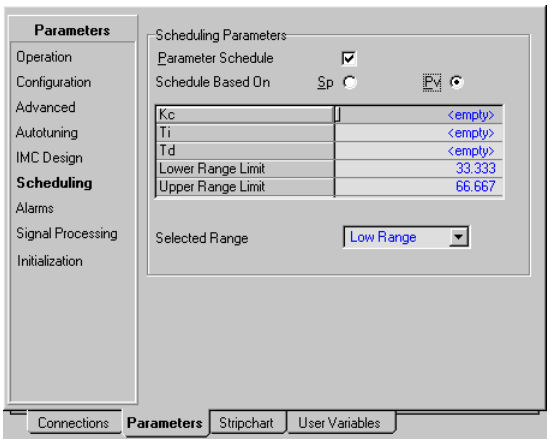
Group	Description
Process Model	Contains the parameters for the process model which are required by the IMC calculator. • Process Gain • Process Time Constraint • Process Delay • Design To
IMC PID Tuning	Displays the operation parameters.

As soon as you enter the parameters in the Process Model group, the operation parameters are calculated and displayed in the IMC PID Tuning group. You can accept them as the current tuning parameters by clicking the Update Tuning button.

Scheduling Page

The Scheduling page gives you the ability to do parameter scheduling. This feature is quite useful for nonlinear processes where the process model changes significantly over the region of operation.

Figure 5.63



The parameter scheduling is activated through the **Parameter Schedule** checkbox. You can use three different sets of PID parameters if you so desire for three different regions of operation. The following regions of operation can be specified from the Selected Range drop-down list.

- Low Range
- Middle Range
- High Range

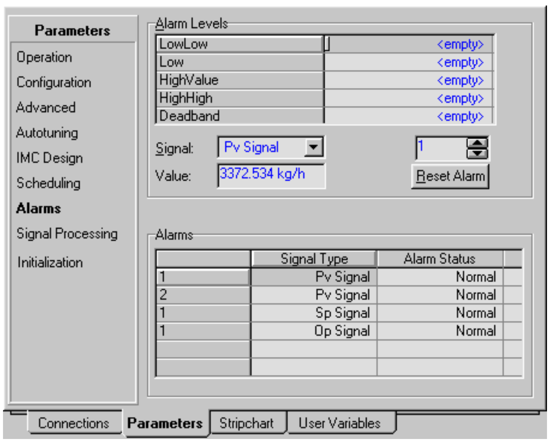
These regions of operations can be based either on the setpoint, or PV of the controller. The ranges can also be specified, the default values are 0-33%, 33%-66%, and 66%-100% of the selected scheduling signal. You need to specify the middle range limit by defining the Upper and Lower Range Limits.

The values of 0 and 100 cannot be specified for both the Lower and the Upper Range Limits.

Alarms Page

The Alarms page allows you to set alarm limits on all exogenous inputs to and outputs from the controller.

Figure 5.64



The Alarms page contains two groups:

- Alarm Levels
- Alarms

Alarm Levels Group

The Alarm Level group allows you to set, and configure the alarm points for a selected signal type. There are four alarm points that could be configured:

- LowLow
- Low
- High
- HighHigh

The alarm points should be specified in the descending order from HighHigh to LowLow points. You cannot specify the value of the Low and LowLow alarm points to be higher than the signal value. Similarly, the High and HighHigh alarm points cannot be specified to a value lower than the signal value. Also, no two alarm points can be specified to a similar value. In addition, you can specify a deadband for a given set of alarms. This can be helpful in situations where the signal is “noisy” to avoid constant triggering of the alarm. If a deadband is specified, you have to specify the alarm points so that their difference is greater than the deadband. At present the range for the allowable deadband is as follows:

$0.0\% \le \text{deadband} \le 1.5\%$ of the signal range.

The above limits are set internally and are not available for adjustment by the user.

Alarms Group

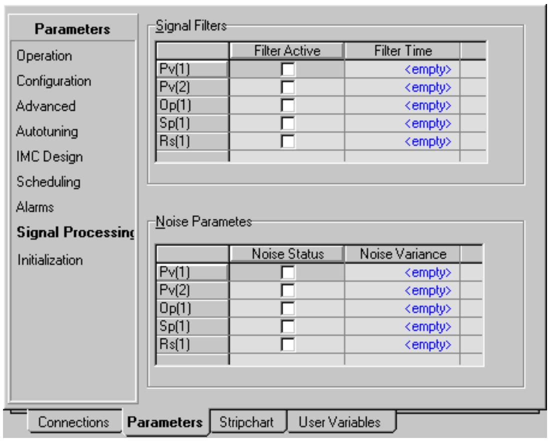
The Alarms group displays the recently violated alarm for the following signals:

Signal	Description
PV	Process Variable
OP	Output
SP	Setpoint

Signal Processing Page

The Signal Processing page allows you to add filters to any signal associated with the operation, as well as test the robustness of any tuning on the controller.

Figure 5.65



This page consists of two groups:

- Signal Filters
- Noise Parameters

Both of these groups allow you to filter, and test the robustness of the following tuning parameters:

- Pv
- Op
- Sp
- Dv
- Rs

To apply the filter select the checkbox corresponding to the signal you want to filter. Once active you can specify the filter time. As you increase the filter time you are filtering out frequency information from the signal. For example, the signal is noisy, there is a smoothing effect noticed on the plot of the PV. Notice that it is possible to add a filter that makes the controller unstable. In such cases the controller needs to be returned. Adding a filter has the same effect as changing the process the controller is trying to control.

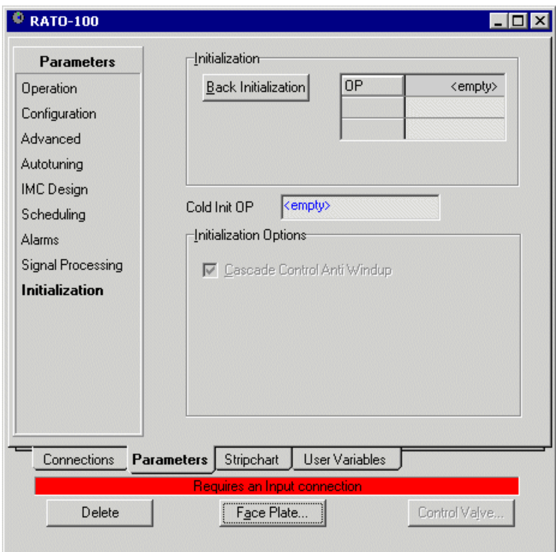
Activating a Noise Parameter is done the same way as adding a filter. However, instead of specifying a filter time you are specifying a variance. Notice that if a high variance on the PV signal is chosen the controller may become unstable. As you increase the noise level for a given signal you observe a some what random variation of the signal.

Initialization Page

The Initialization page allows you to set up a sophisticated controller by taking into account the problem of saturation in cascade control, and the need for an appropriate initial output value to ensure a smooth start-up. The Initialization page consists of two features:

- Back Initialization
- Cascade Control Anti Windup

Figure 5.66



Back Initialization

A proper initial OP Value is supplied to the controller to ensure the integration runs smoothly during start up. The Back Initialization button is used to initialize the controller output based on the current position of the executor (for example, a valve, a stream, or another controller). This prevents disturbances in the system during the initial switch-over.

Cascade Control Anti Windup

For more information on the cascade control strategy, refer to **Section 3.3 - Basic Control** in the **HYSYS Dynamic Modeling** guide.

A common problem associated with cascade control is saturation. Saturation occurs when the primary controller continues to integrate and send out correction signals to the secondary controller even when the output of the secondary controller is already at its designed limit. As a result, when the primary offset changes (decreases or increases), the primary controller cannot respond accordingly until it overcomes the saturation. By the time this happens, the primary offset is once again too large to be adjusted. This severely reduces the controller performance and even creates an unstable system as the output is always fluctuating.

The Cascade Control Anti Windup checkbox allows you to prevent saturation by having the primary controller automatically calculate the feasible output that can be executed by the secondary controller. Once the primary controller detects that the output of the secondary controller has reached its limit (upper or lower), the primary controller will not integrate any further from getting into saturation. Thus, when the offset changes, both the primary and secondary controllers can react immediately without having to wait for the saturation to clear.

Stripchart Tab

Refer to **Section 1.3.7 - Stripchart Page/Tab** for more information.

The Stripchart tab allows you to select and create default strip charts containing various variable associated to the operation.

User Variables Tab

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

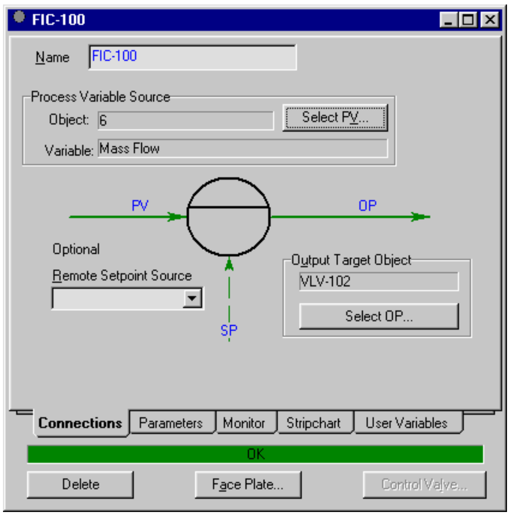
The User Variables tab enables you to create and implement your own user variables for the current operation.

5.4.4 PID Controller

The Controller operation is the primary means of manipulating the model in Dynamic mode. It adjusts a stream (OP) flow to maintain a specific flowsheet variable (PV) at a certain value (SP).

The Controller can cross the boundaries between flowsheets, enabling you to sense a process variable in one flowsheet and control a valve in another.

Figure 5.67



The PID Controller property view contains the following tabs:

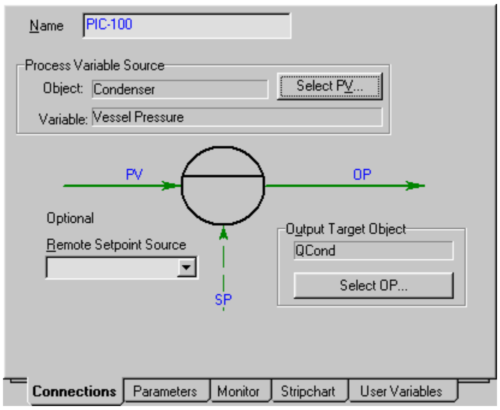
- Connections
- Parameters
- Monitor
- Stripchart
- User Variables

Connections Tab

The Connections tab allows you to select both the PV and OP. It is comprised of six objects described in the table below:

Object	Description
Name	Contains the name of the controller. It can be edited by selecting the field and entering the new name.
Process Variable Source Object	Contains the Process Variable Object (stream or operation) that owns the variable you want to control. It is specified via the Variable Navigator.
Process Variable	Contains the Process Variable you want to control.
Output Target Object	The stream or valve, which is controlled by the PID Controller operation
Select PV/OP	These two buttons open the Variable Navigator which selects the Process Variable and the Output Target Object respectively.
Remote Setpoint Source	If you are using set point from a remote source, select the remote Setpoint Source associated with the Master controller

Figure 5.68



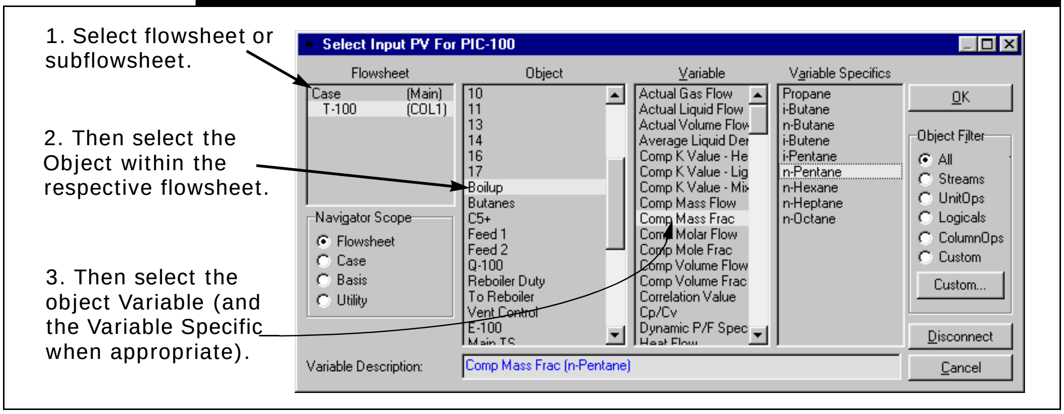
Process Variable Source

Common examples of PVs include vessel pressure and liquid level, as well as stream conditions such as flow rate or temperature.

The Process Variable, or PV, is the variable that must be maintained, or controlled at a desired value.

To attach the Process Variable Source, click the Select PV button. Then select the appropriate object and variable simultaneously, using the Variable Navigator.

Figure 5.69



Refer to **Section 1.3.9 - Variable Navigator Property View** for information.

The Variable Navigator property view allows you to simultaneously select the Object and Variable.

Remote Setpoint Source

The Remote Setpoint Source drop-down list allows you to select the remote sources from a list of existing operations.

A Spreadsheet cell can also be a Remote Setpoint Source.

The "cascade" mode of the controller no longer exits. Instead what is available now is the ability to switch the setpoint from local to remote. The remote setpoint can come from another object such as a spreadsheet, or another controller cascading down a setpoint. In other words, a master in the classical cascade control scheme.

When PID parameters are exported to a PID controller from a HYSYS spreadsheet, the controller gets initialized at each time step.

When the spreadsheet exports any PID parameter (gain, Ti, and/or Td) to a PID controller, the controller calls ControllerInitialization(), which is required for smooth switch when the user change the PID parameters. However, if there is an export variable connected an output object, the spreadsheet updates the output every integration step even if the value has not changed. So with the spreadsheet constantly changing the PID parameter values in every integration step, the PID will not be functioning.

Output Target Object

The Controller compares the Process Variable to the Setpoint, and produces an output signal which causes the manipulated variable to open or close appropriately.

The Output of the Controller is the control valve which the Controller manipulates in order to reach the set point. The output signal, or OP, is the desired percent opening of the control valve, based on the operating range which you define in the Control Valve property view.

Selecting the Output Target Object is done in a similar manner to selecting the Process Variable Source. You are also limited to objects supported by the controller and not currently attached to another controller.

The information regarding the control valve or control op port sizing is contained on a sub-view accessed by clicking the **Control Valve** or **Control OP Port** button found at the bottom of the PID Controller property view.

The Control Valve button (at the bottom right corner of the controller operation property view) appears if the OP is a stream.

The Control OP Port button (at the bottom right corner of the controller operation property view) appears when the OP is not a stream and there a range of specified values is required.

Parameters Tab

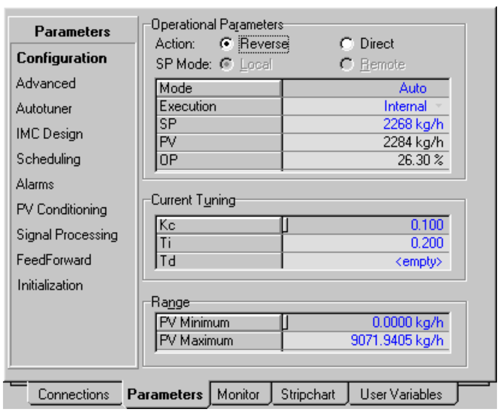
The Parameters tab contains the following pages:

- Configuration
- Advanced
- Autotuner
- IMC Design
- Scheduling
- Alarms
- PV Conditioning
- Signal Processing
- FeedForward
- Initialization

Configuration Page

The Configuration page allows you to set the process variable range, controller action, operating mode, and depending on the mode, either the SP or OP, as well as tune the controller.

Figure 5.70



PV and SP

The PV (or Process Variable) is the measured variable, which the controller is trying to keep at the Setpoint.

The SP (or Setpoint) is the value of the Process Variable, which the Controller is trying to meet. Depending on the Mode of the Controller, the SP is either entered by the user or displayed only.

For the Controller to become operational, you must:

1. Define the minimum and maximum values for the PV (the Controller cannot switch from Off mode unless PVmin and PVmax are defined).
2. Once you provide these values (as well as the Control Valve span), you may select the Automatic mode, and give a value for the Setpoint.

HYSYS uses the current value of the PV as the set point by default, but you may change this value at any time.

Without a PV span, the Controller cannot function.

HYSYS converts the PV range into a 0-100% range, which is then used in the solution algorithm. The following equation is used to translate a PV value into a percentage of the range:

$$PV(\%) = \left(\frac{PV - PV_{min}}{PV_{max} - PV_{min}}\right)100 \qquad (5.16)$$

OP

The OP (or Output) is the percent opening of the control valve. The Controller manipulates the valve opening for the output stream in order to reach the set point. HYSYS calculates the necessary OP using the controller logic in all modes with the exception of Manual. In Manual mode, you may input a value for the output, and the setpoint becomes whatever the PV is at the particular valve opening you specify.

Modes

The Controller operates in any of the following modes:

Controller Mode	Description
Off	The controller does not manipulate the control valve, although the appropriate information is still tracked.
Manual	Manipulates the controller output manually.
Auto	The controller reacts to fluctuations in the Process Variable, and manipulates the output according to the logic defined by the tuning parameters.
Casc	The main controller reacts to the fluctuations in the Process Variable, and sends signals to the slave controller (Remote Setpoint Source).
Indicator	Allows you to simulate the controller without controlling the process.

Refer to **Section 5.13.2 - Controller Face Plate** for more information on Face Plate.

The mode of the controller can also be set on the Face Plate.

Execution

You can select where the signal from the controller is sent using the drop-down list in the Execution field. You have two selections:

- **Internal**. Confines the signals generated to stay within HYSYS.
- **External**. Sends the signals to a DCS, if a DCS is connected to HYSYS.

Tuning

The Tuning group allows you to define the constants associated with the PID control equation. The characteristic equation for a PID Controller is given below:

$$OP(t) = OP_{ss} + K_c E(t) + \frac{K_c}{T_i}\int E(t)\mathrm{d}t + K_c T_d \frac{dE(t)}{dt} \tag{5.17}$$

where:

OP(t) = controller output at time t

OP_{ss} = steady state controller output (at zero error)

$E(t)$ = error at time t

K_c = proportional gain of the controller

T_i = integral (reset) time of the controller

T_d = derivative (rate) time of the controller

The error at any time is the difference between the Setpoint and the Process Variable:

$$E(t) = SP(t) - PV(t) \tag{5.18}$$

Depending on which of the three tuning parameters you have specified, the Controller responds accordingly to the error. A Proportional-only controller is modeled by providing only a value for K_p, while a PI (Proportional-Integral) Controller requires values for K_p and T_i. Finally, the PID (Proportional-Integral-Derivative) Controller requires values for all three of K_p, T_i, and T_d.

Action

There are two options for the Action of the controller, which are described in the table below:

Controller Action	Description
Direct	When the PV rises above the SP, the OP increases. When the PV falls below the SP, the OP decreases.
Reverse	When the PV rises above the SP, the OP decreases. When the PV falls below the SP, the OP increases.

The Controller equation given above applies to a Reverse-acting Controller. That is, when the PV rises above the SP, the error becomes negative and the OP decreases. For a Direct-response Controller, the OP increases when the PV rises above the SP. This action is made possible by replacing *Kp* with *-Kp* in the Controller equation. A typical example of a Reverse Acting controller is in the temperature control of a Reboiler. In this case, as the temperature in the vessel rises past the SP, the OP decreases, in effect closing the valve and hence the flow of heat.

Some typical examples of Direct-Acting and Reverse-Acting control situations are given below.

- Direct - Acting Controller Example 1: Flow Control in a Tee

  Suppose you have a three-way tee in which a feed stream is being split into two exit streams. You want to control the flow of exit stream Product 1 by manipulating the flow of stream Product 2:

Process Variable and Setpoint	Product 1 Flow
Output	Product 2 Flow
When Product 1 Flow rises *above* the SP	The OP increases, in effect increasing the flow of Product 2 and decreasing the flow of Product 1.
When Product 1 Flow falls *below* the SP	The OP decreases, in effect decreasing the flow of Product 2 and increasing the flow of Product 1.

- Direct - Acting Controller Example 2: Pressure Control in a Vessel

  Suppose you were controlling the pressure of a vessel V-100 by adjusting the flow of the outlet vapour, SepVapour:

Process Variable and Setpoint	V-100 Vessel Pressure
Output	SepVapour Flow
When V-100 Pressure rises *above* the SP	The OP increases, in effect increasing the flow of SepVapour and decreasing the Pressure of V-100.
When V-100 Pressure falls *below* the SP	The OP decreases, in effect decreasing the flow of SepVapour and increasing the Pressure of V-100.

- Reverse - Acting Controller Example 1: Temperature Control in a Reboiler

  Reverse-Acting control may be used when controlling the temperature of reboiler R-100 by adjusting the flow of the duty stream, RebDuty:

Process Variable and Setpoint	R-100 Temperature
Output	RebDuty Flow
When R-100 Temperature rises above the SP	The OP decreases, in effect decreasing the flow of RebDuty and decreasing the Temperature of R-100.
When R-100 Temperature falls below the SP	The OP increases, in effect increasing the flow of RebDuty and increasing the Temperature of R-100.

- Reverse - Acting Controller Example 2: Pressure Control in a Reboiler

  Another example where Reverse-Acting control may be used is when controlling the stage pressure of a reboiler R-100 by adjusting the flow of the duty stream, RebDuty:

Process Variable and Setpoint	R-100 Stage Pressure
Output	RebDuty Flow
When R-100 Stage Pressure rises above the SP	The OP decreases, in effect decreasing the flow of RebDuty and decreasing the Stage Pressure of R-100.
When R-100 Stage Pressure falls below the SP	The OP increases, in effect increasing the flow of RebDuty and increasing the Stage Pressure of R-100.

SP Mode

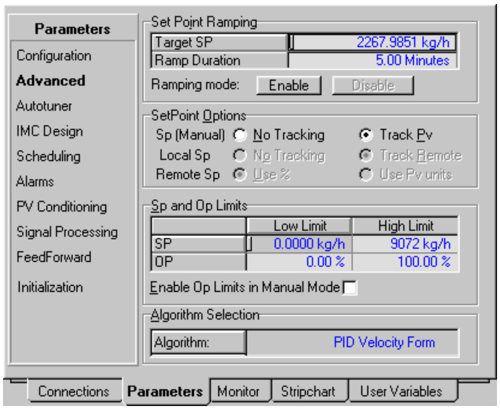
You have the ability to switch the setpoint from local to remote. Essentially, there are two internal setpoints in the controller, the first is the local setpoint where you can manually specify the setpoint via the property view (interface), and the other is the remote setpoint which comes from another object such as a spreadsheet or another controller cascading down a setpoint. In other words, a master in the classical cascade control scheme.

Advanced Page

The Advanced page contains the following four groups:

Group	Description
Set Point Ramping	Allows you to specify the ramp target and duration.
SetPoint Options	Contains the options for setpoint tracking.
Sp and Op Limits	Allows you to set the upper and lower limits for set point and output targets.
Algorithm Selection	Contains the PID controller algorithms for output calculation.

Figure 5.71



Set Point Ramping Group

The setpoint ramping function has been modified in the present PID controllers. Now it is continuous (in other words, when enabled by clicking the **Enable** button), the setpoint changes over the specified period of time in a linear manner.

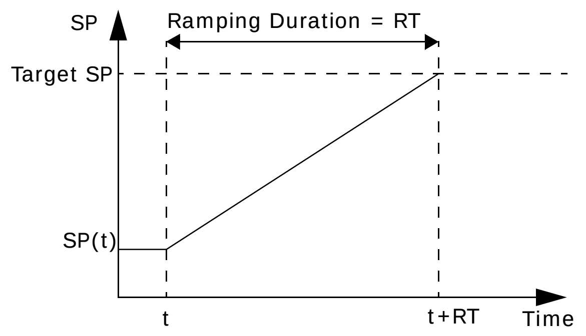
Setpoint ramping is only available in Auto mode.

The Set Point Ramping group contains the following two fields:

- **Target SP**. Contains the Setpoint you want the Controller to have at the end of the ramping interval. When the ramping is disabled, the Target SP field displays the same value as the SP field on the Configuration page.

- **Ramp Duration**. Contains the time interval you want to complete setpoint change in.

Figure 5.72



Controller Ramping

There are also two buttons available in this group:

- **Enable**. Activates the ramping process
- **Disable**. Stops the ramping process.

While the controller is in ramping mode, you can change the setpoint as follows:

- Enter a new setpoint in the Target SP field, on this page.
- Enter a new setpoint in the SP field, on the Configuration page.

During the setpoint ramping the Target SP field, shows the final value of the setpoint whereas the SP field, on the Configuration page, shows the current setpoint seen internally by the control algorithm.

During ramping, if a second setpoint change has been activated, then Ramping Duration time would be restarted for the new setpoint.

An example, if you click the Enable button and enter values for the two parameters in the Set Point Ramping group, the Controller switches to Ramping mode and adjust the Setpoint linearly (to the Target SP) during the Ramp Duration, see **Figure 5.72**.

For example, suppose your current SP is 100, and you want to change it to 150. Rather than creating a sudden, large disruption by manually changing the SP while in Automatic mode, click the Enable button and enter an SP of 150 in the Target SP input cell. Make the SP change occur over, say, 10 minutes by entering this time in the Ramp Duration cell. HYSYS adjusts the SP from 100 to 150 linearly over the 10 minute interval.

SetPoint Options Group

In the past the PID controllers implemented an automatic setpoint tracking in manual mode, in other words, the value of the setpoint was set equal to the value of the PV when the controller was placed in manual mode. This meant that upon switching, the values of the setpoint and PV were equal, and therefore there was an automatic bumpless transfer.

In the present controller setup, the Sp (Manual) option allows PV tracking, by selecting the No Tracking radio button, when the controller is in manual mode. However, when the controller is switched into the automatic mode from manual, there is an internal resetting of the controller errors to ensure that there is an instantaneous bumpless transfer prior to the controller recognizing a setpoint that is different from the PV.

If the Track PV radio button is selected than there would be an automatic setpoint tracking.

The Local Sp option allows you to disable the tracking for the local setpoint when the controller is placed in manual mode. You can also have the local setpoint track the remote setpoint by selecting the Track Remote radio button.

The Remote Sp option allows you to select either the **Use %** radio button (for restricting the setpoint changes to be in percentage) or **Use Pv units** radio button (for setpoint changes to be in PV units).

- **Use %**. If this radio button is selected, then the controller reads in a value in percentage from a remote source and uses the PV range to calculate the new setpoint.

- **Use Pv units**. If this radio button is active, then the controller reads in a value from a remote source, and is used as the new setpoint. The remote sources setpoint must have the same units as the controller PV.

An example, it is desired to control the flowrate in a stream with a valve. A PID controller is used to adjust the valve opening to achieve the desired flowrate, that is set to range between 0.2820 m^3/h and 1.75 m^3/h. A spreadsheet is used as a remote source for the controller setpoint. A setpoint change to 1 m^3/h from the current Pv value of 0.5 m^3/h is made. The spreadsheet internally converts the new setpoint as m^3/s (in other words, 1/3600 = 0.00028 m^3/s) and pass it to the controller, which converts it back into m^3/h (in other words, 1 m^3/h). The controller uses this value as the new setpoint. If the units are not specified, then the spreadsheet passes it as 1 m^3/s, which is the base unit in HYSYS, and the controller converts it into 3600 m^3/h and pass it on to the SP field as the new setpoint. Since the PV maximum value cannot exceed 1.75m^3/h, the controller uses the maximum value (**1.75 m^3/h**) as the new setpoint.

Sp and Op Limits Group

This group enables you to specify the output and setpoint limits. The output limits ensure that a predetermined minimum or maximum output value is never exceeded. In the case of the setpoint, the limits enforce an acceptable the range of values that could be entered via the interface or from a remote source.

When the **Enable Op Limits in Manual Mode** checkbox is selected, you can enable the set point and output limits when in manual mode.

Algorithm Selection Group

In the Algorithm Selection group, you can select one of the three available controller update algorithms:

- PID Velocity Form
- PID Positional Form (ARW = Anti-Reset Windup)
- PID Positional Form (noARW)
- PID Manual Loading

Velocity or Differential Form

In the velocity or differential form the controller equation is given as:

$$u(t) = u(t-1) + K_c\left[e(t) - e(t-1) + \frac{1}{T_i}e(t)h + T_d\frac{(e(t) - 2e(t-1) + e(t-2))}{h}\right] \tag{5.19}$$

where:

u(t) = controller output and t is the enumerated sampling instance in time

u(t-1) = value of the output one sampling period ago

K_c, T_i, and T_d = controller parameters

h = sampling period

The velocity or differential form of the controller should be applied when there is an integral term. When there is no integral term a positional form of the controller should be used.

Positional Form

In the positional form of the algorithm, the controller output is given by:

$$u(t) = K_c\left[e(t) + \frac{1}{T_i}\sum_{i=1}^{n} e(i)h + T_d\frac{(e(t) - e(t-1))}{h}\right] \tag{5.20}$$

Here it is important to handle properly the summation term associated with the integral part of the control algorithm. Specifically, the integral term could grow to a very large value in instances where the output device is saturated, and the PV is still not able to get to the setpoint. For situations like the one above, it is important to reset the value of the summation to ensure that the output is equal to the limit (upper or lower) of the controller output. As such, when the setpoint is changed to a region where the controller can effectively control, the controller responds immediately without having to decrease a summation term that has grown way beyond the upper or lower limit of the output. This is referred to as an automatic resetting of the control integral term commonly called anti-reset windup.

In HYSYS, both algorithms are implemented as presented above with one key exception, there is no derivative kick. This means that the derivative part of the control algorithm operates on the process variable as opposed to the error term. As such the control equation given in **Equation (5.19)** is implemented as follows:

$$u(t) = u(t-1) + K_c\left[e(t) - e(t) + \frac{1}{T_i}e(kh) + T_d\frac{(-pv + 2pv(t-1) - pv(t-2))}{h}\right] \quad (5.21)$$

Manual Loading Station

In the manual loading station algorithm the output, $u(t)$, is equal to the input $y(t)$.

$$u(t) = y(t) \quad (5.22)$$

In Manual mode, you can set the OP like regular PID.
In Auto mode, the OP equals to PV based on PV range.

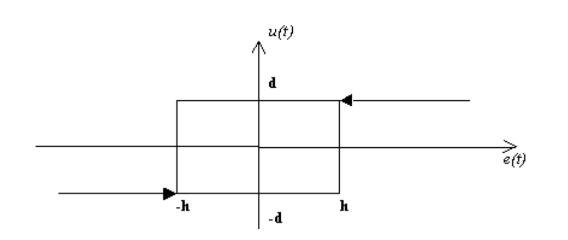
Here the setpoint plays no role in the algorithm, instead whatever the percent input value is going into the controller, the output follows the same percent value. As in the case of the other control algorithms the output is bounded by an upper and lower limit of 100% and 0.0% respectively.

Autotuner Page

The autotuner function provides tuning parameters for the PID controller based on gain and phase margin design. The autotuner itself can be viewed as another controller object that has been embedded into the PID controller. The autotuner is based on a relay feedback technique, and by default incorporates a relay with hysteresis (h).

The figure below shows an example of a relay with an amplitude (d) and hysteresis (h) is plotted on a graph of *Output* ***u(t)*** versus *Error Input into the relay* ***e(t)*** plot.

Figure 5.73

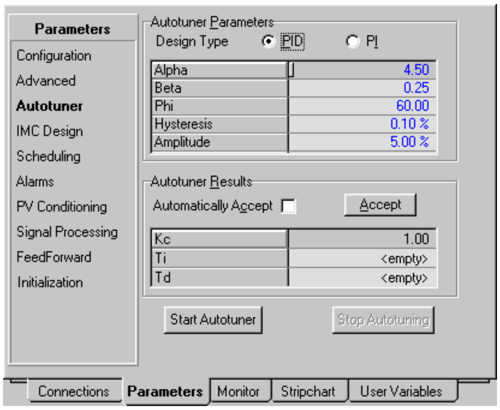


This type of relay is a double-valued nonlinearity, sometimes referred to as having memory. In other words, the value of the output depends on the direction that the process error is coming. Relays are quite common in automation and control, and this technique for tuning PID controllers has been around at least 10 years now (see Cluett and Goberdhansingh, Automatica, 1992). The technique has a strong theoretical base and in general works well in practice but it is not a panacea.

The PID controller parameters that are obtained from the autotuner are based on a design methodology that makes use of a gain margin at a specified phase angle. This design is quite similar to the regular gain and phase margin methodology except that it is more accurate since the relay has the ability to determine points in the frequency domain accurately and quickly. Also, the relay experiment is controlled and does not take a long time during the tuning cycle.

The Autotuner page allows you to specify the autotuning parameters.

Figure 5.74



The page contains two groups:

- **Autotuner Parameters**. Contains the parameters required by the Autotuner to calculate the controller parameters.
- **Autotuner Results**. Displays the resulting controller parameters. You have the option to accept the results as the current tuning parameters

Autotuner Parameters Group

In this group, you can specify the controller type by selecting the PID or PI radio button for the Design Type. In the present autotuner implementation there are four parameters that the you must specify which are as follows:

Parameter	Range
Ratio (Ti/Td) (Alpha)	$3.0 \le \alpha \le 6.0$
Gain ratio (Beta)	$0.10 \le \beta \le 1.0$
Phase angle (Phi)	$30° \le \phi \le 65°$
Relay hysteresis (h)	$0.01\% \le h \le 5.0\%$
Relay amplitude (d)	$0.5\% \le d \le 10.0\%$

In the present version of the software there are default values specified for the PID tuning. Before starting the autotuner, you must ensure that the controller is in the manual or automatic mode, and the process is relatively steady.

If you move the cursor over the tuning parameters field, the Status Bar displays the parameters range.

Autotuner Results Group

This group displays the results of the autotuner calculation, and allows you to accept the results as the current controller setting. The **Start Autotuner** button activates the tuning calculation, and the **Stop Autotuning** button abort the calculations.

After running the autotuner, you have the option to accept the results either automatically or manually. Selecting the **Automatically Accept** checkbox sets the resulting controller parameters as the current value instantly. If the Automatically Accept checkbox is inactive, you can specify the calculated controller parameters to be the current setting by clicking the Accept button.

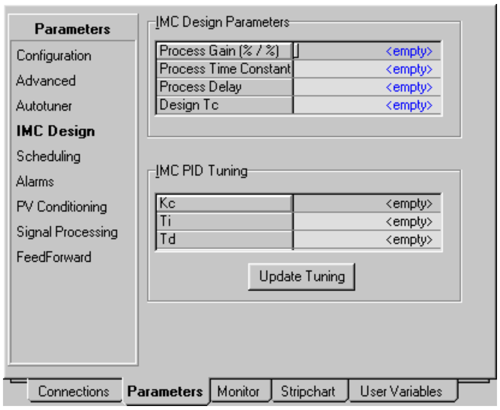
An example, while a case is running in a dynamic simulation, change the controller mode to either Manual or Automatic. On the **Autotuner** page, select the Design Type and specify the tuning parameters (or use the default values). Click the Start Autotuner button and wait for the Autotuner to display the results. To accept the results and copy them in the Current Tuning group on the Configuration page, click the Accept button.

HYSYS suggest using the auto tuning results as a guideline and should not be treated as a catholicon. It is recommended to specify the Autotuning parameters to suit your process requirement.

IMC Design Page

The IMC Design page allows you to use the internal model control (IMC) calculator to calculate the PID parameters based on a specified model of the process one is attempting to control.

Figure 5.75



The IMC method is quite common in most of the process industries and has a very solid theoretical basis. In general, the performance obtained using this design methodology is superior to most of the existing techniques for tuning PIDs. As such, when there is a process model available (first order plus delay) this approach should be used to determine the controller parameters.

You must specify a design time constant, which is usually chosen as three times that of the measured process time constant. The IMC Design page has the following two groups:

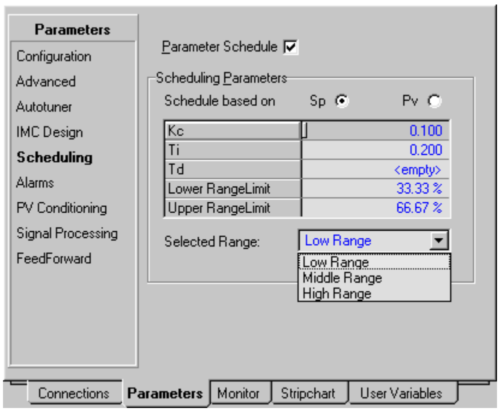
Group	Description
IMC Design Parameters	Contains the parameters for the process model which are required by the IMC calculator. • Process Gain • Process Time Constraint • Process Delay • Design To
IMC PID Tuning	Displays the PID controller parameters.

As soon as you enter the parameters in the IMC Design Parameters group, the controller parameters are calculated and displayed in the IMC PID Tuning group. You can accept them as the current tuning parameters by clicking the Update Tuning button.

Scheduling Page

The Scheduling page gives you the ability to do parameter scheduling.

Figure 5.76



The parameter scheduling is quite useful for nonlinear processes where the process model changes significantly over the region of operation. The parameter scheduling is activated through the **Parameter Schedule** checkbox. You can use three different sets of PID parameters, if you so desire for three different regions of operation. The following regions of operation can be specified from the Selected Range drop-down list.

- Low Range
- Middle Range
- High Range

These regions of operations can be based either on the setpoint or PV of the controller. The ranges can also be specified, the default values are 0-33%, 33%-66%, and 66%-100% of the selected scheduling signal.

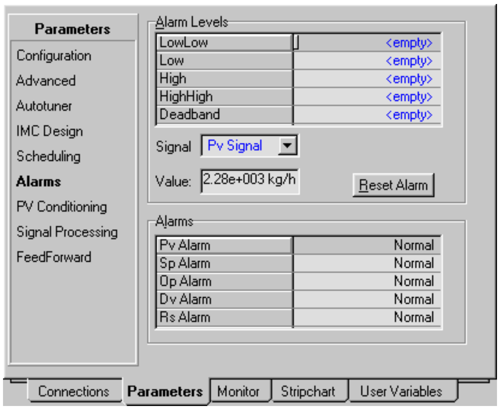
You need to specify the middle range limit by defining the Upper and Lower Range Limits.

> **The values of 0 and 100 cannot be specified for both the Lower and the Upper Range Limits.**

Alarms Page

The Alarms page allows you to set alarm limits on all exogenous inputs to and outputs from the controller.

Figure 5.77



The page contains two groups and one button:

- Alarm Levels
- Alarms
- Reset Alarm button

Alarm Levels Group

The Alarm Level group allows you to set, and configure the alarm points for a selected signal type. There are four alarm points that can be configured:

- LowLow
- Low
- High

- HighHigh

The alarm points should be specified in the descending order from HighHigh to LowLow points. You cannot specify the value of the Low and LowLow alarm points to be higher than the signal value. Similarly, the High and HighHigh alarm points cannot be specified a value lower than the signal value. Also, no two alarm points can have a similar value. In addition, you can specify a deadband for a given set of alarms. This can be helpful in situations where the signal is “noisy” to avoid constant triggering of the alarm. If a deadband is specified, you have to specify the alarm points so that their difference is greater than the deadband. At present the range for the allowable deadband is as follows:

$0.0\% \leq \text{deadband} \leq 1.5\%$ of the signal range.

The above limits are set internally and are not available for adjustment by the user.

Alarms Group

The Alarms group display the recently violated alarm for the following signals:

Signal	Description
PV	Process Variable
OP	Output
SP	Setpoint
DV	Disturbance Variable (this is available for the feedforward controller in the Future)
RS	Remote Setpoint

Reset Alarm Button

When the deadband has been set, it is possible that an alarm status is triggered and the alarm does not disappear until the band has been exceeded. The Reset Alarm button allows the alarm to be reset when within the deadband.

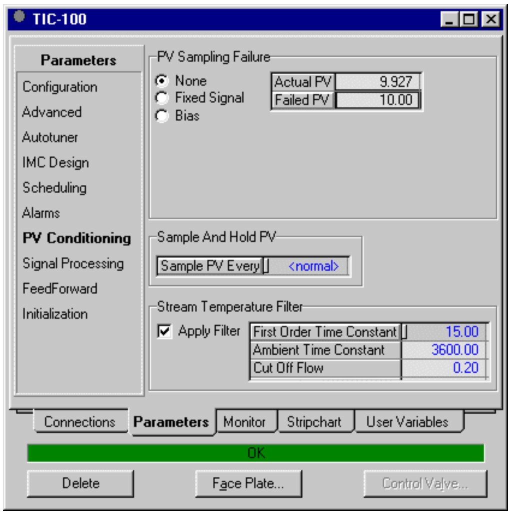
An example, it is desired to control the flowrate through a valve within the operating limits. Multiple alarms can be set to alert you about increases or decreases in the flowrate. For the purpose of this example, you are specifying low and high alarm limits for the process variable signal. Assuming that the normal flowrate passing through the valve is set at 1.2 m^3/h, the low alarm should get activated when the flowrate falls below 0.7 m^3/h. Similarly, when the flowrate increases to 1.5 m^3/h the high alarm should get triggered.

To set the low alarm, first make sure that the Pv Signal is selected in the Signal drop-down list. Specify a value of 0.7 m^3/h in the cell beside the Low alarm level. Follow the same procedure to specify a High alarm limit at 1.5 m^3/h. If you want to re-enter the alarms, click the Reset Alarm button to erase all the previously specified alarms.

PV Conditioning Page

The PV Conditioning page allows you to simulate the failure of the controller input signal.

Figure 5.78



This page consists of three groups:

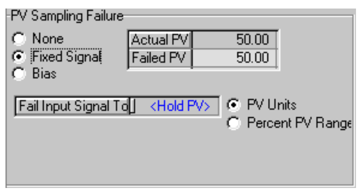
- PV Sampling Failure
- Sample and Hold PV
- Stream Temperature Filter

PV Sampling Failure Group

The PV Sampling Failure group consists of three radio buttons: None, Fixed Signal, and Bias. The options presented to you changes with respect to the radio button chosen.

- When the None radio button is selected, the property view is as seen in the figure above, with only the Actual PV and Failed PV values displayed.
- When Fixed Signal radio button is selected, the PV Sampling Failure group appears as follows.

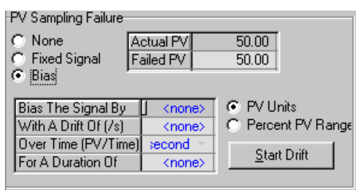
Figure 5.79



The Failed Input Signal To parameter allows you to fix the failed input signal using either the PV units or a Percentage of the PV range.

- When the Bias radio button is selected the PV Sampling Failure group appears as follows.

Figure 5.80



The PV Sampling Failure group allows you to drift the input signal. The parameters allow you to bias the signal and create a drift over a period of time. To start the drift simply click the Start Drift button.

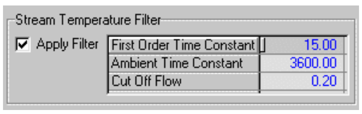
Sample and Hold PV Group

The Sample And Hold PV group allows you to take a PV sample and hold this value for a specified amount of time.

Stream Temperature Filter Group

The Stream Temperature Filter group allows you to calculate the temperature of a low flow rate stream by applying a first order transient filter with a user-specified ambient time constant.

Figure 5.81



By default, the **Apply Filter** checkbox is cleared. You can apply the temperature filter by selecting the **Apply Filter** checkbox. To set the conditions for the filter, you will need to specify the following:

- **First Order Time Constant**. First order exponential time constant applied to the filter when the flow rate is within the acceptable range (in other words, above the Cut Off Flow value). By default, the First Order Time Constant is 15 seconds. If the field is empty, or you enter a value of zero, then no filtering is applied.
- **Ambient Time Constant**. Time constant applied to the filter when the flow rate of a stream drops below the Cut Off Flow value. It determines how long it takes for the actual temperature of the stream to reach the ambient temperature. By default, the ambient time constant is 3600 seconds. If the field is empty, or you enter a value of zero, the temperature value is calculated from the flash and no filtering is applied.
- **Cut Off Flow**. Switch-over point at which the temperature filter applies the ambient time constant in calculating the temperature of the stream. The Cut Off Flow value is expressed in molar flow, and the default value is 1e-5 kmol/s.

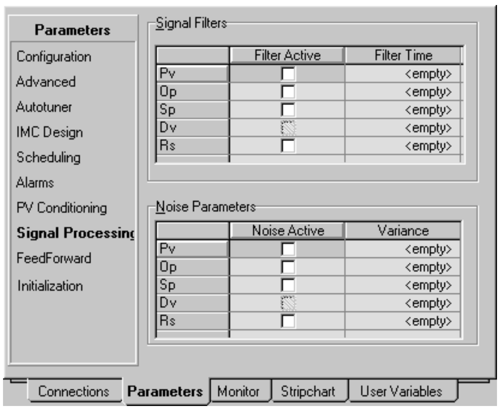
If the stream flow rate is above the Cut Off Flow value, the controller automatically switches back to the normal flash calculations which only apply the First Order Time Constant. The

Ambient Time Constant is applied when the flow rate drops below the Cut Off Flow value with the temperature ramping to ambient over some slow periods.

Signal Processing Page

The Signal Processing page allows you to add filters to any signal associated with the PID controller, as well as test the robustness of any tuning on the controller.

Figure 5.82



This page is made up of two groups: Signal Filters and Noise Parameters. Both of these groups allows you to filter and test the robustness of the following tuning parameters:

- Pv
- Op
- Sp
- Dv
- Rs

To apply the filter, select the checkbox corresponding to the signal you want to filter. Once active you can specify the filter time. As you increase the filter time you are filtering out frequency information from the signal. It is possible to add a filter that makes the controller unstable. In such cases the controller needs to be returned. Adding a filter has the same effect as changing the process the controller is trying to control.

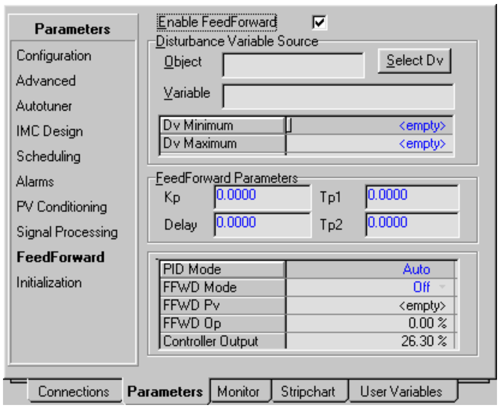
For example, if the signal is noisy, there is a smoothing effect noticed on the plot of the PV when the filter is applied.

Activating a Noise Parameter is done the same way as adding a filter. However, instead of specifying a filter time you are specifying a variance. Notice that if a high variance on the PV signal is chosen the controller may become unstable. As you increase the noise level for a given signal you observe a some what random variation of the signal.

FeedForward Page

The FeedForward page enables you to design a controller that takes into account measured disturbances.

Figure 5.83



To enable feedforward control you must select the Enable FeedForward checkbox.

The Disturbance Variable Source group allows you to select a disturbance variable, and minimum and maximum variables. The disturbance variable is specified by clicking the Select Dv button. This opens the Variable Navigator.

The FeedForward Parameters group allows you to set the Operating Mode for both the PID controller and the FeedForward controller and tune the controller.

All FeedForward controllers require a process model in order for the controllers to work properly. Presently HYSYS uses a model that results in a lead-lag process. Therefore, there are four parameters available.

The equation model for the FeedForward controller is as follows:

$$G(s) = K_p \frac{(\tau_1 s + 1)e^{-ds}}{\tau_2 s + 1} \tag{5.23}$$

where:

K_p = *gain*

τ = *time constant*

d = *deadtime or delay*

Initialization Page

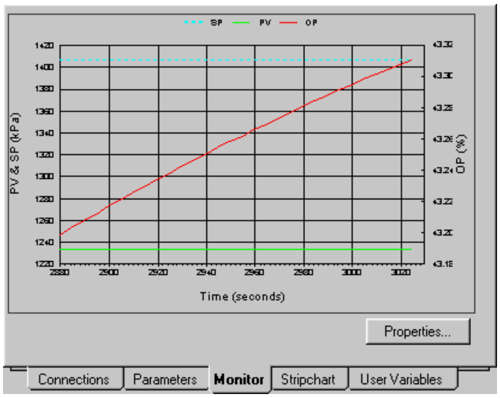
For more information on back initialization and saturation, refer to **Section 5.4.3 - Ratio Controller**.

The Initialization page of the PID controller contains the same information as the one for the ratio controller.

Monitor Tab

A quick monitoring of the response of the Process Variables, Setpoint, and Output can be seen on the Monitor tab. This tab allows you to monitor the behaviour of process variables in a graphical format while calculations are proceeding.

Figure 5.84

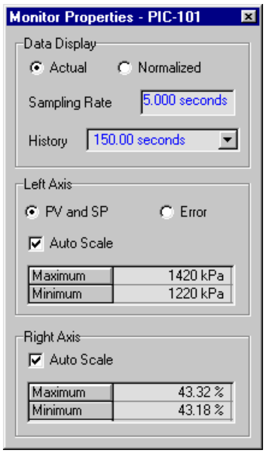


The Monitor tab displays the PV, SP, and Op values in their relevant units versus time. You can customize the default plot settings using the object inspection menu, which is available only when you right-click on any spot on the plot area.

The object inspection menu contains the following options:

Item	Description
Graph Control	Opens the **Graph Control Property View** to modify many of the plot characteristics.
Turn Off/On Cross Hair	Turns the cross hair either on or on.
Turn Off/On Vertical Cross Hair	Turns the vertical cross hair either on or on.
Turn Off/On Horizontal Cross Hair	Turns the horizontal cross hair either on or on.
Values Off/On	Displays the current values for each of the variables, if turned on.
Copy to Clipboard	Copies the current plot to the clipboard with the chosen scale size.

Item	Description
Print Plot	Prints the plot as it appears on the screen.
Print Setup	Allows you to access the typical Windows Print Setup. The Windows Print Setup allows you to select the printer, the paper orientation, the paper size, and paper source.



A quick way to customize your plot is to use the Monitor Properties property view, which can be access by clicking the Properties button.

There are three group available on the Monitor Properties property view, which are described as follows:

- **Data Capacity**. Allows you specify the type and amount of data to be displayed. You can also select the data sampling rate.
- **Left Axis**. Gives you an option to display either the PV and SP or the Error data on the left axis of the plot. You can also customize the scale or let HYSYS auto scale it according to the current values.
- **Right Axis**. Gives you an option to either customize the right axis scale or let HYSYS auto scale it according to the current OP value.

Stripchart Tab

Refer to **Section 1.3.7 - Stripchart Page / Tab** for more information.

The Stripchart tab allows you to select and create default strip charts containing various variable associated to the operation.

User Variables Tab

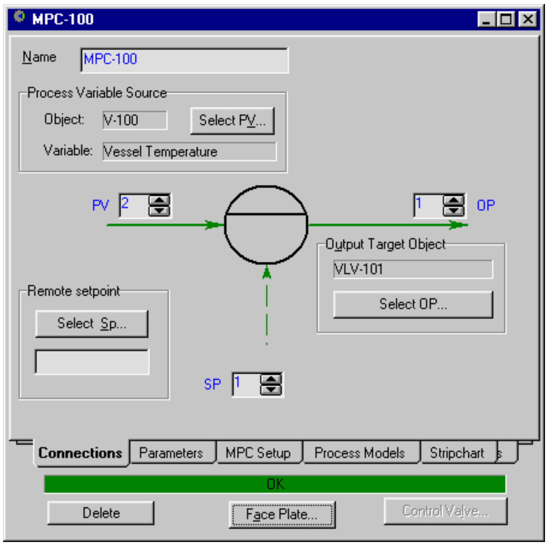
For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables tab enables you to create and implement your own user variables for the current operation.

5.4.5 MPC Controller

The “Model Predictive Control” (MPC) controller addresses the problem of controlling processes that are inherently multi-variable and interacting in nature, in other words, one or more inputs affects more than one output.

Figure 5.85



The current version of the MPC implementation does not handle the problem of processes with constraints - a future release is capable of handling that class of problems.

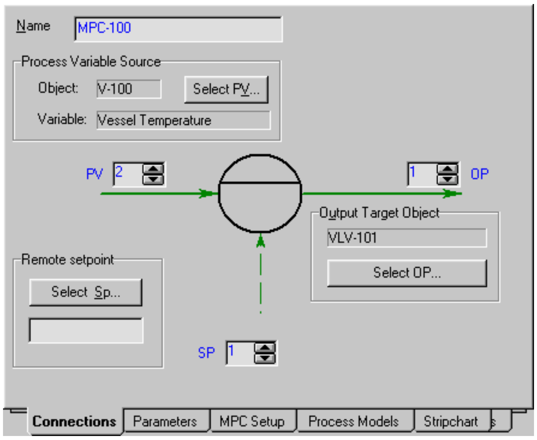
The MPC property view contains the following tabs:

- Connections
- Parameters
- MPC Setup
- Process Models
- Stripchart
- User Variables

Connections Tab

The Connections tab is comprised of six objects that allow you to select the Process Variable Source, Output Target Object, and Remote SP.

Figure 5.86



Object	Description
Name	Contains the name of the controller. It can be edited by selecting the field and entering the new name.
Process Variable Source Object	Contains the Process Variable Object (stream or operation) that owns the variable you want to control. It is specified via the Variable Navigator.
Process Variable	Contains the Process Variable you want to control.
Output Target Object	The stream or valve which is controlled by the PID Controller operation
Remote Setpoint Source	If you are using set point from a remote source, select the remote Setpoint Source associated with the Master controller
Select PV / OP / SP	These three buttons open the Variable Navigator which selects the Process Variable, the Output Target Object, the Remote Setpoint Source respectively.
PV / OP / SP	These three fields allow you to select a specific Process Variable, Output Target Object, and Remote Setpoint Source respectively.

Process Variable Source

Common examples of PVs include vessel pressure and liquid level, as well as stream conditions such as flow rate or temperature.

The Process Variable, or PV, is the variable that must be maintained or controlled at a desired value.

Refer to **Section 1.3.9 - Variable Navigator Property View** for information.

To attach the Process Variable Source, click the **Select PV** button. Then select the appropriate object and variable simultaneously, using the Variable Navigator. The Variable Navigator allows you to simultaneously select the Object and Variable.

Remote Setpoint

The “cascade” mode of the controller no longer exits. Instead what is available now is the ability to switch the setpoint from local to remote. The remote setpoint can come from another object such as a spread-sheet or another controller cascading down a setpoint. In other words, a master in the classical cascade control scheme.

A Spreadsheet cell can also be a Remote Setpoint Source.

The Select Sp button allows you to select the remote source using the Variable Navigator.

Output Target Object

The Controller compares the Process Variable to the Setpoint, and produces an output signal which causes the manipulated variable to open or close appropriately.

The Output of the Controller is the control valve which the Controller manipulates in order to reach the set point. The output signal, or OP, is the desired percent opening of the control valve, based on the operating range which you define in the Control Valve property view.

Selecting the Output Target Object is done in a similar manner to selecting the Process Variable Source. You are also limited to objects supported by the controller and not currently attached to another controller.

The information regarding the control valve or control op port sizing is contained on a sub-view accessed by clicking the **Control Valve** or **Control OP Port** button found at the bottom of the MPC Controller property view.

The Control Valve button (at the bottom right corner of the logical operation property view) appears if the OP is a stream.

The Control OP Port button (at the bottom right corner of the logical operation property view) appears when the OP is not a stream and there a range of specified values is required.

Parameters Tab

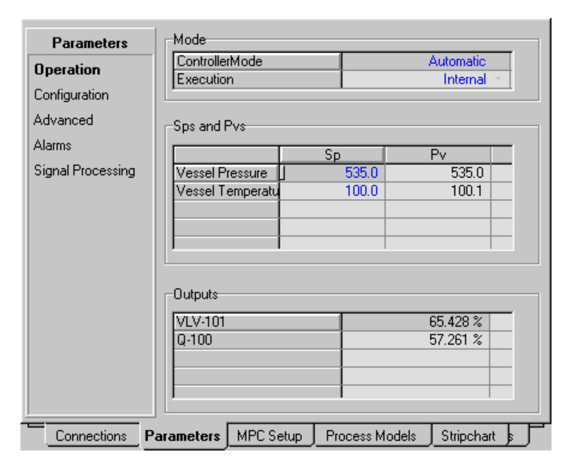
The Parameters tab contains the following pages:

- Operation
- Configuration
- Advanced
- Alarms

Operation Page

The Operation page allows you to set the execution type, controller mode and depending on the mode, either SP or OP.

Figure 5.87



Mode Group

The mode of the controller may also be set on the Face Plate, refer to **Section 5.13.2 - Controller Face Plate** for more information.

The Controller operates in any of the following modes:

- **Off**. The Controller does not manipulate the control valve, although the appropriate information is still tracked.
- **Manual**. Manipulates the Controller output manually.
- **Automatic**. The Controller reacts to fluctuations in the Process Variable and manipulates the Output according to the logic defined by the tuning parameters.

You can select where the signal from the controller is sent using the drop-down list in the **Execution** field. You have two selections:

- **Internal**. Confines the signals generated to stay within HYSYS.
- **External**. Sends the signals to a DCS, if a DCS is connected to HYSYS.

Sps and Pvs Group

Displays the Setpoint (SP) and Process Variable (PV) for each of the controllers inputs. Depending on the Mode of the controller the SP can either be input by you or is determined by HYSYS.

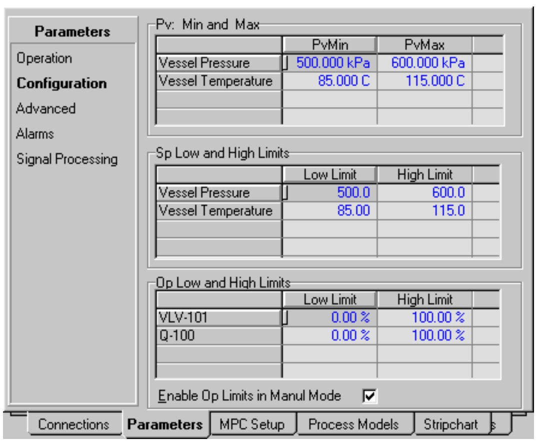
Outputs Group

The Output (OP) is the percent opening of the control valve. The Controller manipulates the valve opening for the Output Stream in order to reach the set point. HYSYS calculates the necessary OP using the controller logic in all modes with the exception of Manual. In Manual mode, you may enter a value for the Output, and the Setpoint becomes whatever the PV is at the particular valve opening you specify. This can be done for all of the inputs to the controller.

Configuration Page

The Configuration page allows to specify the process variable, setpoint, and output ranges.

Figure 5.88



PV: Min and Max

For the Controller to become operational, you must:

1. Define the minimum and maximum values for the PV (the Controller cannot switch from Off mode unless PVmin and PVmax are defined).
2. Once you provide these values (as well as the Control Valve span), you can select the Automatic mode and give a value for the Setpoint.

HYSYS uses the current value of the PV as the set point by default, but you can change this value at any time.

Without a PV span, the Controller cannot function.

HYSYS converts the PV range into a 0-100% range, which is then used in the solution algorithm. The following equation is used to translate a PV value into a percentage of the range:

$$PV(\%) = \left(\frac{PV - PV_{min}}{PV_{max} - PV_{min}}\right)100 \tag{5.24}$$

SP Low and High Limits

You can specify the higher and lower limits for the Setpoints to reflect your needs and safety requirements. The Setpoint limits enforce an acceptable range of values that could be entered via the interface or from a remote source. By default the PVs min and max values are used as the SPs low and high limits, respectively.

Op Low and High Limits

You can specify the higher and lower limits for all the outputs. The output limits ensure that a predetermined minimum or maximum output value is never exceeded. By default 0% and 100% is selected as a low and a high of limit, respectively for all the outputs.

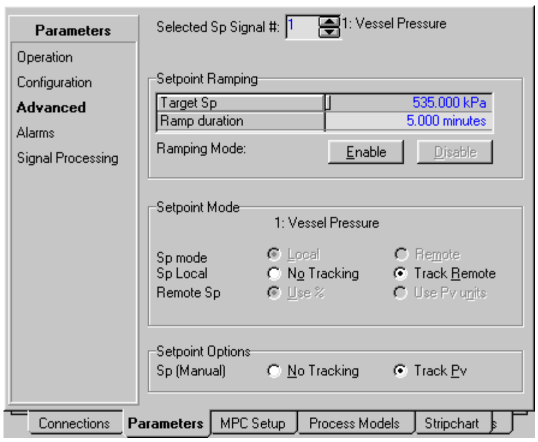
When the Enable Op Limits in Manual Mode checkbox is selected, you can enable the set point and output limits when in manual mode.

Advanced Page

The Advanced page contains the following three groups:

Group	Description
Setpoint Ramping	Allows you to specify the ramp target and duration.
Setpoint Mode	Contains the options for setpoint mode and tracking, as well as the option for remote setpoint.
Setpoint Options	Contains the option for setpoint tracking only in manual mode.

Figure 5.89



The setpoint signal is specified in the **Selected Sp Signal #** field by clicking the up or down arrow button, or by typing the appropriate number in the field.

Depending upon the signal selected, the page displays the respective setpoint settings.

Setpoint Ramping Group

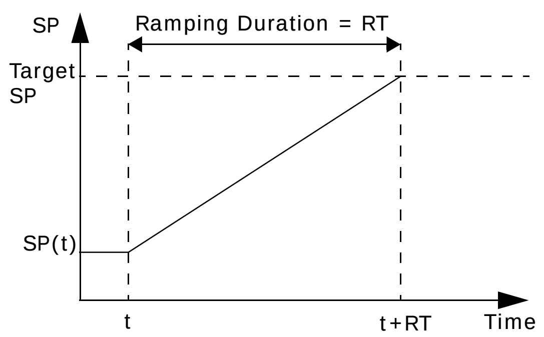
The setpoint ramping function has been modified in the present MPC controllers. Now it is continuous, in other words, when set to on by clicking the Enable button, the setpoint changes over the specified period of time in a linear manner.

Setpoint ramping is only available in Auto mode.

The Setpoint Ramping group contains the following two fields:

- **Target SP**. Contains the Setpoint you want the Controller to have at the end of the ramping interval. When the ramping is turn off, the Target SP field display the same value as SP field on the Configuration page.
- **Ramping Duration**.Contains the time interval you want to complete setpoint change in.

Figure 5.90



Controller Ramping

Besides these two fields there are also two buttons available in this group:

- **Enable**. Activates the ramping process.
- **Disable**. Stops the ramping process.

While the controller is in ramping mode, you can change the setpoint as follows:

- Enter a new setpoint in the Target SP field.
- Enter a new setpoint in the SP field, on the Operation page.

During the setpoint ramping the Target SP field shows the final value of the setpoint whereas the SP field, on the Operation page, shows the current setpoint seen internally by the control algorithm.

During ramping, if a second setpoint change has been activated, then Ramping Duration time would be restarted for the new setpoint.

An example, if you click the Enable button and enter values for the two parameters in the Setpoint Ramping group, the Controller switches to Ramping mode and adjusts the Setpoint linearly (to the Target SP) during the Ramp Duration, see **Figure 5.90**. For example, suppose your current SP is 100, and you want to change it to 150. Rather than creating a sudden large disruption by manually changing the SP while in Automatic mode, click the Enable button and enter the SP of 150 in the Target SP input field. Make the SP change occur over, say, 10 minutes by entering this time in the Ramp Duration field. HYSYS adjusts the SP from 100 to 150 linearly over the 10 minute interval.

Setpoint Mode Group

You now have the ability to switch the setpoint from local to remote using the Setpoint mode radio buttons. Essentially, there are two internal setpoints in the controller, the first is the local setpoint where you can manually specify the setpoint via the property view (interface), and the other is the remote setpoint which comes from another object such as a spreadsheet or another controller cascading down a setpoint. In other words, a master in the classical cascade control scheme.

The Sp Local option allows you to disable the tracking for the local setpoint when the controller is placed in manual mode. You can also have the local setpoint track the remote setpoint by selecting the Track Remote radio button.

The Remote Sp option allows you to select either the **Use%** radio button (for restricting the setpoint changes to be in percentage) or **Use Pv units** radio button (for setpoint changes to be in Pv units).

- **Use%**. If the Remote Sp is set to Use%, then the controller reads in a value in percentage from a remote source, and using the Pv range calculates the new setpoint.
- **Use Pv unit**s. If the Remote Sp is set to Use Pv units, then the controller reads in a value from a remote source and sets a new setpoint. The remote source's setpoint must have the same units as the controller Pv.

SetPoint Options Group

If the Track PV radio button is selected, then there is automatic setpoint tracking in manual mode, that sets the value of the setpoint equal to the value of the Pv prior to the controller being placed in the manual mode. This means that upon switching from manual to automatic mode the values of the setpoint and Pv were equal and, therefore, there was an automatic bumpless transfer. Also you have the option not to track the pv, by clicking the **No Tracking** radio button, when the controller is placed in manual mode. However, when the controller is switched into the automatic mode from manual, there is an internal resetting of the controller errors to ensure that there is an instantaneous bumpless transfer prior to the controller recognizing a setpoint that is different from the Pv.

An example, it is desired to control the flowrate in a stream with a valve. A MPC controller is used to adjust the valve opening to achieve the desired flowrate, that is set to range between 0.2820 m^3/h and 1.75 m^3/h. A spreadsheet is used as a remote source for the controller setpoint. A setpoint change to 1 m^3/h from the current PV value of 0.5 m^3/h is made. The spreadsheet internally converts the new setpoint as m^3/s (**1/3600 = 0.00028 m^3/s**) and passes it to the controller, which reads the value and converts it back into m^3/h (**1 m^3/h**). The controller uses this value as the new setpoint. If the units are not specified, then the spreadsheet passes it as 1 m^3/s, which is the base unit in HYSYS, and the controller converts it into 3600 m^3/h and passes it on to the SP field as the new setpoint. Since the

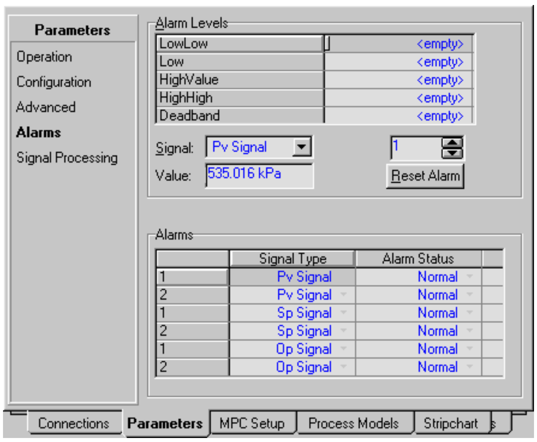
PV maximum value cannot exceed 1.75m^3/h, the controller uses the maximum value (**1.75 m^3/h**) as the new setpoint.

Alarms Page

The Alarms page allows you to set alarm limits on all exogenous inputs to and outputs from the controller. The page contains two groups:

- Alarm Levels
- Alarms

Figure 5.91



Alarm Levels Group

The Alarm Level group allows you to set, and configure the alarm points for a selected signal type. There are four alarm points that could be configured:

- LowLow
- Low
- High
- HighHigh

The alarm points should be specified in the descending order from HighHigh to LowLow points. You cannot specify the value of the Low and LowLow alarm points to be higher than the signal value. Similarly, the High and HighHigh alarm points cannot be specified a value lower than the signal value. Also, no two alarm points can be specified a similar value. In addition, you can specify a deadband for a given set of alarms. This can be helpful in situations where the signal is “noisy” to avoid constant triggering of the alarm. If a deadband is specified, you have to specify the alarm points so that their difference is greater than the deadband. At present, the range for the allowable deadband is as follows:

$0.0\% \leq \text{deadband} \leq 1.5\%$ of the signal range.

The above limits are set internally and are not available for adjustment by the user.

Alarms Group

The Alarms group displays the recently violated alarm for the following signals:

Signal	Description
PV	Process Variable
OP	Output
SP	Setpoint

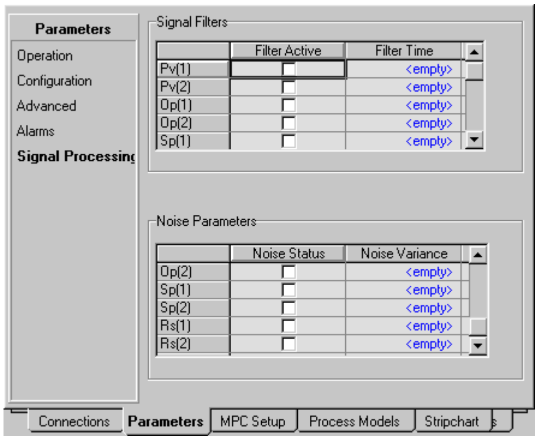
An example, it is desired to control the flowrate through a valve within the operating limits. These limits can be monitored using the Alarms feature in MPC Controller. Multiple alarms can be set to alert you about increase or decrease in the flowrate. For the purpose of this example, you are specifying low and high alarm limits for the process variable signal. Assuming that the normal flowrate passing through the valve is set at 1.2 m^3/h, the low alarm should get activated when the flowrate falls below 0.7 m^3/h. Similarly, when the flowrate increases to 1.5 m^3/h the high alarm should get triggered.

To set the low alarm, first make sure that the Pv Signal is selected in the Signal drop-down list. Specify a value of 0.7 m^3/h in the cell beside the Low alarm level. Follow the same procedure to specify a High alarm limit at 1.5 m^3/h. If you want to re-enter the alarms, click the Reset Alarm button to erase all the previously specified alarms.

Signal Processing Page

The Signal Processing page allows you to add filters to any signal associated with the MPC controller, as well as test the robustness of any tuning on the controller.

Figure 5.92



This page is made up of two groups:

- Signal Filters
- Noise Parameters

Both of these groups allow you to filter, and test the robustness of the following tuning parameters:

- Pv
- Op
- Sp
- Rs

To apply the filter select the checkbox corresponding to the signal you want to filter. Once active you can specify the filter time. As you increase the filter time you are filtering out frequency information from the signal. For example the signal is noisy, there is a smoothing effect noticed on the plot of the PV.

Notice that it is possible to add a filter that makes the controller unstable. In such cases the controller needs to be returned. Adding a filter has the same effect as changing the process the controller is trying to control.

Activating a Noise Parameter is done the same way as adding a filter. However, instead of specifying a filter time you are specifying a variance. Notice that if a high variance on the PV signal is chosen the controller may become unstable. As you increase the noise level for a given signal you observe a some what random variation of the signal.

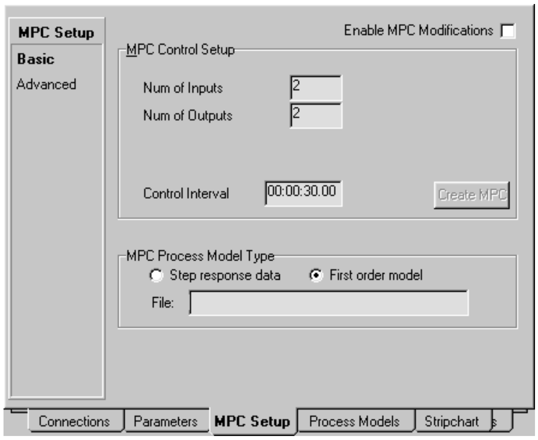
MPC Setup Tab

The MPC controller has a number of setup options available. These options are available on the Basic and Advanced pages of the Setup tab. In order to change any of the default values specified on these pages it is necessary to enable the MPC modifications checkbox. Whatever the option chosen, it is important to establish a sampling period (control interval) first. Specifically, the sampling period must be chosen to be consistent with the **sampling theorem** (see Shannon's Sampling Theorem). As such, it should be about 1/5 to 1/10 of the smallest time-constants. If the process is heavily dominated by process deadtime then the sampling period should be based on the deadtime. In situations where the process models are a mix of fast and slow process dynamics care should be taken in selecting the sampling period. A carefully designed MPC controller is an effective and efficient controller.

Basic Page

The Basic page divides the setup settings into MPC Control Setup and MPC Process Model Type groups.

Figure 5.93



MPC Control Setup Group

In the MPC Control Setup group you are required to specify the following:

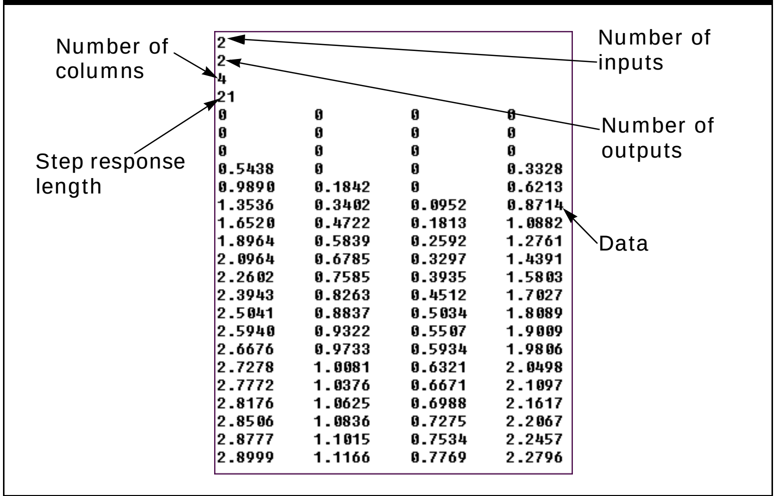
- **Num of Inputs**. Allows you to specify the number of process input. Up to a maximum of 12 process inputs can be specified. The default value is 1.
- **Num of Outputs**. Allows you to specify the number of process output. Up to a maximum of 12 process inputs can be specified. The default value is 1.
- **Control Interval**. Allows you to specify the control or sampling interval. The default value is 30 seconds.

Anytime one of the MPC setting is changed, a new MPC object has to be created internally-this is automatically achieved by clicking on the Create MPC button.

MPC Process Model Type Group

You have the option to specify the model to be either Step response data or a First order model. If the Step response data radio button is selected, then a text file can be used to input the process model. The input file must follow a specific format in terms of inputs and outputs, as well as columns of data. The following is a description of the ASCII text file required for the input:

Figure 5.94

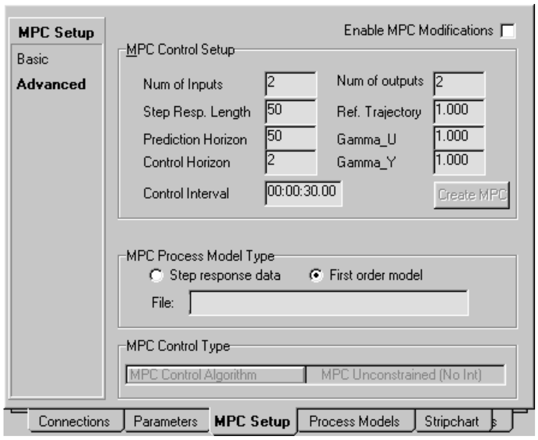


The step response data is typically obtained either directly from plant data, or they are deducted from other so-called parametric model forms such as Discrete State-Space and Discrete Transfer Function Models.

Advanced Page

The Advanced page divides the setup settings into MPC Control Setup, MPC Process Model Type, and MPC Control Type groups.

Figure 5.95



MPC Control Setup Group

Anytime one of the MPC setting is changed, a new MPC object has to be created internally-this is automatically achieved by clicking on the Create MPC button.

In the MPC Control Setup group you are required to specify the following:

Field	Description
Num of Inputs	The number of process input. Up to a maximum of 12 process inputs can be specified. The default value is 1.
Num of Outputs	The number of process output. Up to a maximum of 12 process outputs can be specified. The default value is 1.
Control interval	The control or sampling interval. The default value is 30 seconds.
Step Resp. length	The number of sampling intervals that is necessary to reach steady state when an input step is applied to the process model. The range of acceptable values are from 15 to 100. The default value is 50.

Field	Description
Prediction Horizon	Allows you to specify how far into the future the predictions are made when calculating the controller output. The Prediction Horizon should be less than or equal to the Step Response Length. The default value is 25.
Control horizon	The number of control moves into the future that are made to achieve the final setpoint. A small control horizon generally means a less aggressive controller. The Control Horizon should be less than or equal to the Prediction Horizon. The default value is 2.
Reference trajectory	The time constant of a first order filter that operates on the true setpoint. A small reference trajectory lets the controller see a pure step as the setpoint is changed. The default value is 1.
Gamma_U/ Gamma_Y	The positive-definite weighting functions, which are associated with the optimization problem that is solved to produce the controller output every control interval. The value of Gamma_U and Gamma_Y should be between 0 and 1. The default value is 1.

Step Response Length

This value represents the number of sampling intervals that is necessary to reach steady state when an input step is applied to the process model. You should consider all of the process models and the sampling interval when selecting step response length. At present, the maximum step response is limited to 100 sampling intervals. Also, the fact that you are specifying the process models in terms of step response means that you are only considering stable processes in this MPC design.

Prediction Horizon and Control Horizon

The prediction horizon represents how far into the future the controller makes its predictions, based on the specified process model. The prediction horizon is limited to the length of the step response, and should be greater than the minimum process model delay. A longer prediction horizon means that the controller looks further into the future when solving for the controller outputs. This may be better if the process model is accurate. In general, you want to take full advantage of the process model by using longer predictions.

The control horizon is the number of control moves into the future the controller considers when making its predictions. In general, the larger the number of moves, the more aggressive the controller is. As a rule of thumb a control horizon of less than 3 is used quite often.

Sampling Interval and reference trajectory

Once you have determined the control interval, other parameters like reference trajectory can be chosen. This value affects the reference setpoint of the predictions used by the MPC problem when solving for the control outputs. Essentially, the reference trajectory represents the time constant of a first order filter that operates on the true setpoint. Hence, a very small value for the reference trajectory implies that the setpoint used in the MPC calculations are close to the actual setpoint. The minimum value for the reference trajectory that can be selected is 1second.

One of the problems that could arise in setting this value “too large” is that the final setpoint reference value, which is used in the predictions, would not be seen by the control algorithm in a given iteration. Therefore, it is important that the reference trajectory value be chosen such that the time constant is smaller than the smallest time constant of the user specified process model set. At present, there is a limit placed on the reference trajectory that is based on the sampling interval and the maximum step-response. However, you should use the process model set as a guide when selecting this value.

In the present version the limits for the reference trajectory is as follows:

$$1 \le \text{Reference Trajectory} \le 15 \times \text{Sampling Interval} \tag{5.25}$$

MPC Process Model Type Group

You have the option to specify the model to be either Step response data or First order model. If the Step response data is selected, then a text file can be used to input the process model. The input file must follow a specific format in terms of inputs and outputs, as well as columns of data, as shown in **Figure 5.94**.

MPC Control Type Group

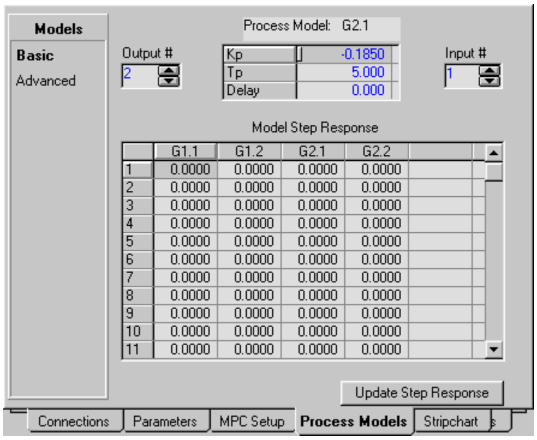
This group allows you to select the MPC Control Algorithm that is used by the controller. At Present, the only option available for selection is the MPC Unconstrainted (No Int). This algorithm does not consider constraints on either controlled and manipulated variables.

Process Models Tab

The Process Models tab allows you to either view the step response data, or specify the first order model parameters.

Basic Page

Figure 5.96



Step Response Data

If the Step response data radio button is selected on the MPC Setup tab, the Process Model tab displays the Model Step Response matrix.

You cannot modify the model step response data on the Process Model tab.

Depending on the number of inputs (i) and outputs (o) the system's dynamics matrix should be an $i \times o$ matrix. The number of process models is equal to the number of outputs or controlled variables. If the Step response data is selected, then the First order model parameters fields are greyed out.

First Order Model

If the First order model is selected on the MPC Setup tab, the Process Model table appears.

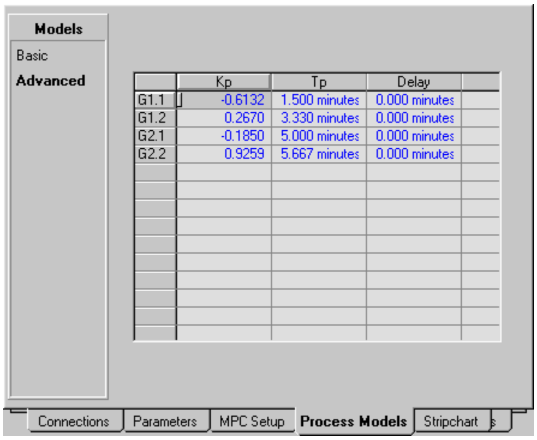
You can specify the first order model parameters for each of the process models, as follows:

1. Select the input and output variable number in the **Input #** and **Output #** selection field by clicking the up or down arrow button , or by typing the appropriate number in the field.
2. Depending upon the input and output variable selected, the relevant process model appears.
3. Then specify the process gain (Kp), process time constant (Tp) and delay for the selected process model in the available matrix.
4. Repeat step# 1-2 for the remaining process models.
5. Then click the **Update Step Response** button to calculate the step response data for the process models.

Advanced Page

The Advanced page lists all of the Process Models, and their associated tuning parameters in table.

Figure 5.97



Stripchart Tab

Refer to **Section 1.3.7 - Stripchart Page / Tab** for more information.

The Stripchart tab allows you to select and create default strip charts containing various variable associated to the operation.

User Variables Tab

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables tab enables you to create and implement your own user variables for the current operation.

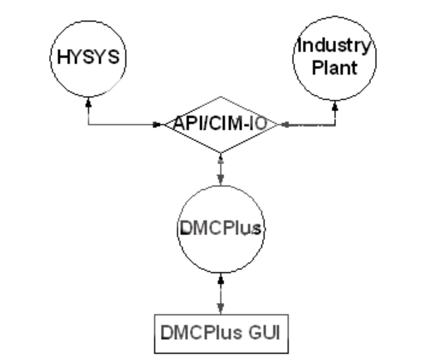
5.4.6 DMCplus Controller

The DMCplus Controller engine runs in Aspen DMCplus Online. HYSYS communicates to DMCplus using the DMCplus API. You are required to have the following licenses to run DMCplus in HYSYS:

- DMCplus Link
- DMCplus Online
- Cim-IO Kernel
- ACO Base

The figure below shows the HYSYS and DMCplus connection. HYSYS works like a Real Plant.

Figure 5.98



Refer to the **Aspen Manufacturing Suite Installation Guide** for information on installing DMCplus Online and DMCplus Desktop.

For information on adding the DMCplus Controller, refer to **Section 5.4.1 - Adding Control Operations**.

You must install DMCplus Online and DMCplus Desktop for the DMCplus Controller to operate properly.

- DMCplus Online is required to run the DMCplus Controller.
- DMCplus Desktop allows you to configure the model used by the DMCplus Controller.

The DMCplus Desktop allows you to configure the models. Each DMCplus Controller requires a Model File (MDL) and a Controller Configuration File (CCF) to operate properly with the Aspen DMCplus.

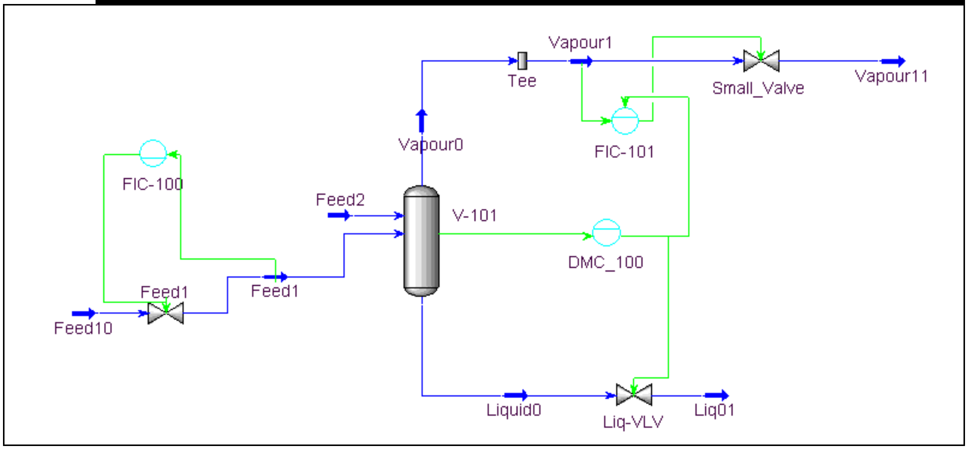
- The MDL file determines the size of the control problem and all the dynamic relationships between each independent and dependent variables.

- The CCF file determines where the input and output parameters for the controller will reside, which optional capabilities will be used, and the values assigned to all of its parameters such as limits, switches, and tuning.

HYSYS can generate the MDL file automatically or record the independent and dependent variable in the collection files (CLC), which contains model testing data. The data in the CLC files are used by the DMCplus Model to create the MDL file. The MDL file is then used by the Aspen DMCplus Build to create a CCF file.

You can add the DMCplus Controller to an existing HYSYS simulation case or to a new HYSYS simulation case that you have created.

Figure 5.99



If the DMCplus Controller is not loaded, it appears in yellow in the HYSYS flowsheet. The status bar on the DMCplus Controller property view will also appear in yellow and indicate that the DMCplus has not been loaded.

The DMCplus property view contains the following tabs:

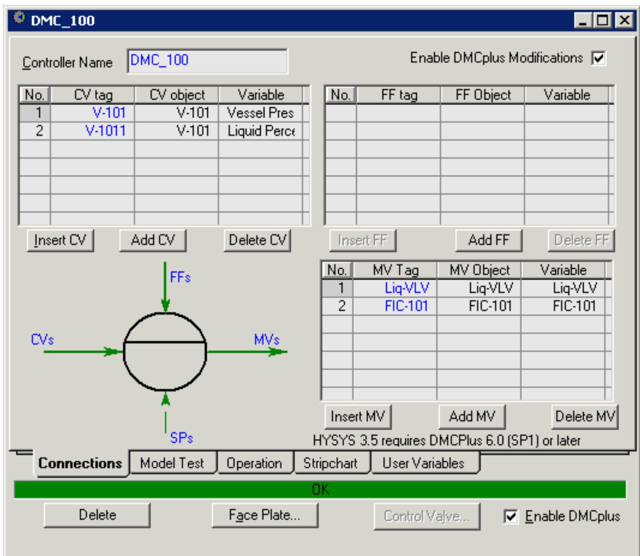
- Connections
- Model Test
- Operation
- Stripchart
- User Variables

The DMCplus Controller property view also contains an **Enable DMCplus** checkbox at the bottom right corner. This checkbox allows you to enable or disable the DMCplus Controller. When the Enable DMCplus checkbox is disabled, the model testing features can be enabled in the **Model Test** tab.

Connections Tab

The Connections tab allows you to define and edit the Controlled, Manipulated and Feed Forward variables for the plant model. You can specify the name of the DMCplus Controller in the Controller Name field.

Figure 5.100



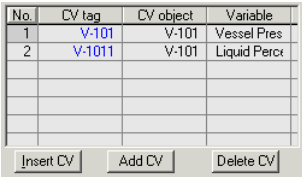
You have to select the **Enable DMCplus Modifications** checkbox to add and edit the Controlled, Manipulated, and Feed Forward variables.

When editing the Controlled and Manipulated variables in the DMCplus Model, ensure that the variable order is the same as in HYSYS.

Controlled Variable (CV)

You must specify a controlled variable for the DMCplus Controller. The controlled variables are the dependent variables that will be controlled by the DMCplus controller.

Figure 5.101



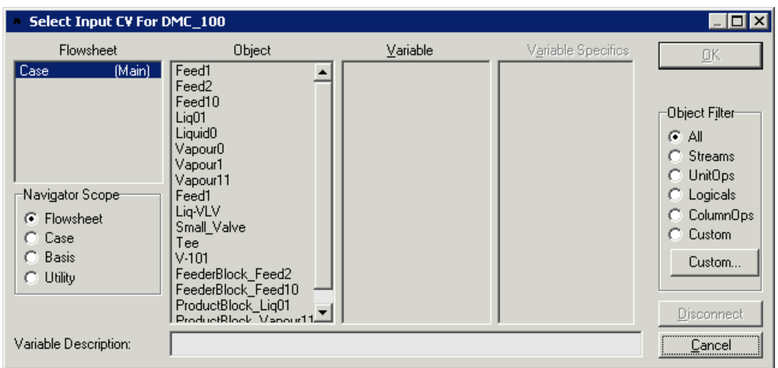
The CV table contains the stream or operation that owns the variable you want to control. The stream or operation is specified using the Select Input property view, which appears when you click the Add CV button or Insert CV button.

- The Add CV button adds the stream or operation after the last defined stream or operation in the table.
- The Insert CV button adds the stream or operation before the currently selected stream or operation in the table.

Refer to **Section 1.3.9 - Variable Navigator Property View** for information on the Select Input property view.

You can select the appropriate object and variable simultaneously using the Select Input property view.

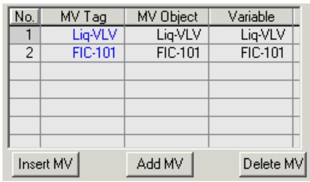
Figure 5.102



Manipulated Variable (MV)

You must specify a manipulated variable for the DMCplus Controller. The manipulated variables are the independent variables that will be manipulated by the DMCplus controller.

Figure 5.103

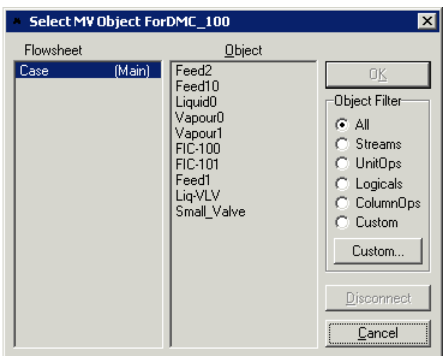


The MV table contains the stream or operation which is controlled by the controller operation.

- The Add MV button adds the stream or operation after the last defined stream or operation in the table.
- The Insert MV button adds the stream or operation before the currently selected stream or operation in the table.

The stream or operation is specified using the Select Object property view, which appears when you click the Add MV button or Insert MV button.

Figure 5.104



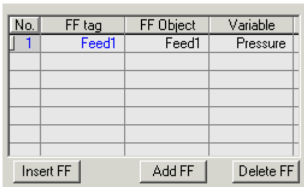
Refer to **Section 5.4.7 - Control Valve** for more information.

When you add a stream to the MV table the Control Valve button is enabled.

Feed Forward (FF)

You can add the Feed Forward variables if needed.

Figure 5.105



The variable takes into account measured disturbances which you can view on the Feed Forward page of the Operation tab. You can add the Feed Forward variables using the Select Input property view similar to when you are adding a controlled variable.

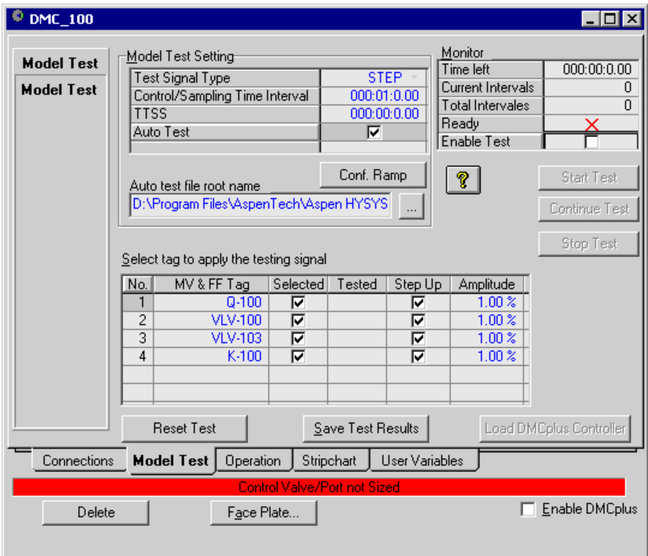
- The Add FF button adds the stream or operation after the last defined stream or operation in the table.
- The Insert FF button adds the stream or operation before the currently selected stream or operation in the table.

The Feed Forward variables are used as independent variables in the DMCplus Model, and they cannot be manipulated by the DMCplus Controller.

Model Test Tab

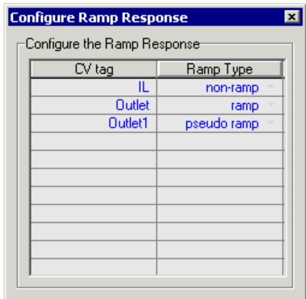
The Model Test tab allows you to set up the DMCplus Controller for model testing. The Model Test page is the only page available on this tab.

Figure 5.106



The following table lists and describes the objects available in the Model Test tab:

Object	Description
Model Test Setting group	
Test Signal Type cell	Enables you to select the signal type for the DMCplus model test. There are two types of signal to choose from: • **PRBS** is simple to use for model identification. • **STEP** is more recognized in practical process applications.
Control/Sampling Time Interval cell	Enables you to specify the amount of time between recorded data points during the testing phase.
TTSS cell	Enables you to specify the total time period available for the model testing. The value should at least be larger than the setting time of the system.

Object	Description
Auto Test checkbox	Enables you to toggle between activating or deactivating the Auto Test option. This option performs test for each of the selected Manipulated and Feed Forward variables one by one. The test results are used to generate an MDL file (which contains the DMCplus controller model). If this checkbox is clear, you need to manually save the testing data to CLC files.
Conf. Ramp button	Enables you to access the Configure Ramp Response property view.  The Configure Ramp Response property view enables you to select the ramp type for each CV tag using the drop-down list available under the **Ramp Type** column. There are three types of ramp type to choose from: • non-ramp • ramp • pseudo ramp The **Conf. Ramp** button is only available if you selected the **Auto Test** option.
Auto test file root name field	Enables you to specify the location and name of the testing data files containing the auto test results and the date and time when the test was performed. This field is only available if the **Auto Test** checkbox is selected.
Epsilon icon 	Enables you to access the File Selection for Saving Test results property view and select the location to save the test result files. This icon is only available if the **Auto Test** checkbox is selected.
Monitor table	
Time left cell	Displays the amount of time left in the model testing.
Current Interval cell	Displays the current interval value during the model testing calculation.

Object	Description
Total Intervals cell	Displays the total number of intervals required to complete the model testing.
Ready cell	Displays an icon to indicate whether the selected model is ready for testing: • Red cross  indicates the model is not ready. • Green checkmark  indicates the model is ready for testing.
Enable Test cell	Enables you to activate the model testing when the **Start Test** button is clicked.
Model Test Help icon 	Enables you to access the help property view that displays the steps required to develop the DMCplus controller.
Start Test button	Enables you to first reset the test and then start the model testing.
Continue Test button	Enables you to continues the last test if it has not been finished.
Stop Test button	Enables you to stop the model testing before the test is complete
Select tag to apply the testing signal table	
No. column	Displays an integer number for each MV and FF variables in the DMCplus controller.
MV & FF Tag column	Displays the name of the MV and FF variables in the DMCplus controller. If a Feed Forward variable (FF) is a dependent (or calculated) variable, HYSYS cannot perform the test at the default/current setting.
Selected column	Contains checkboxes that enable you to toggle between selecting or ignoring the variables for the test. By default, all the Manipulated variables (MV) and Feed Forward variables (FF) are selected to apply the test signal.
Tested column	Display the status of the variable, whether it has been tested or not, during the testing process.
Step Up column	Contains checkboxes that enable you to toggle between step up testing or step down testing for each variable. By default, the checkboxes are set to step up.
Amplitude column	Enables you to specify the percentage value of testing signal amplitude for each variable. By default, the signal amplitude is set at 1.00%.
Reset Test button	Enables you to reset the model testing back to the beginning.

Object	Description
Save Test Results button	Enables you to save the test data results in a *.clc file or a set of *.rec files.
Load DMC Controller button	Enables you to load and run the configured DMCplus controller model in the HYSYS simulation case. This button is disabled if you do not have the configuration (.ccf) and model (.mdl) files in the correct directory.

Performing DMCplus Model Testing

After selecting the controlled, manipulated, and feed forward variables, data needs to be generated from the plant model to develop the controller model.

Follow the steps below to develop the controller model:

1. From the Model Testing Setting group, select the test setting parameters.
   - In the **Test Signal Type** drop-down list, select **STEP** or **PRBS**.
   - In the **Control/Sampling Time Interval** cell, specify the time used to determine how often the data points are recorded during the testing phase.
   - In the **TTSS** cell, specify the total time period during which to apply the testing.
   - In the **Auto Test** cell, use the checkbox to toggle between activating or deactivating the Auto Test option.
   - Click the **Conf. Ramp** button to access the Configure Ramp Response property view. In the Configure Ramp Response property view, select the ramp characteristic/ option for each CV data using the drop-down list under the **Ramp Type** column.
   - In the **Auto test file root name** field, specify the location and name for the generated testing data files. This field is only available if the **Auto Test** checkbox is selected.
2. The **Select tag to apply the testing signal** table contains several options to configure the variable to be tested.

It is recommended that the default setting in the Select tag to apply the testing signal table be modified by advance users for special case.

3. In the **Monitor** table, select the **Enable Test** checkbox, to automatically activate the model testing when the **Start Test** button is clicked.
4. Click the **Start Test** button to start the testing.

> **Before you start the testing, ensure that all the related slave PID Controllers are set to the remote mode.**

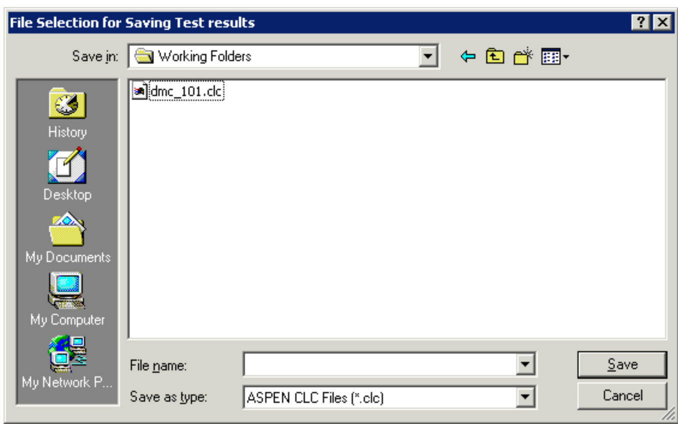
When the **Time left** field displays zero, the testing is finished.

- If you had selected the **Auto Test** option, you can skip steps #5 to #6 because HYSYS will automatically generate an MDL file.
- If you did not select the **Auto Test** option, you have to manually save the test results in a file.

5. Click the **Save Test Result** button, and save the testing results to a ***.clc** file or a set of ***.rec** files.

The CLC or REC file(s) can be processed by the Aspen DMCplus Model to generate a MDL file.

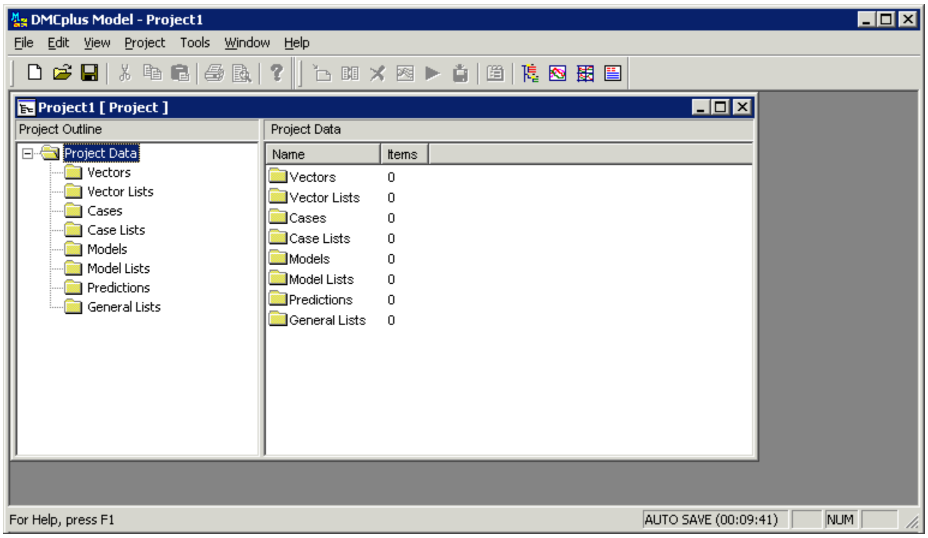
Figure 5.107



Refer to the DMCplus Desktop manual for details.

6. Start the Aspen Model application, load the saved data and build the DMCplus model (***.mdl**).

Figure 5.108



In the DMCplus Model you will import the Collect file (*.clc) to vectors by saving the project first and then adding the CLC file.

When adding the independent (MV) and dependent (CV) variables to a case, ensure that the order you are adding the variables is the same as on the Connections tab in HYSYS.

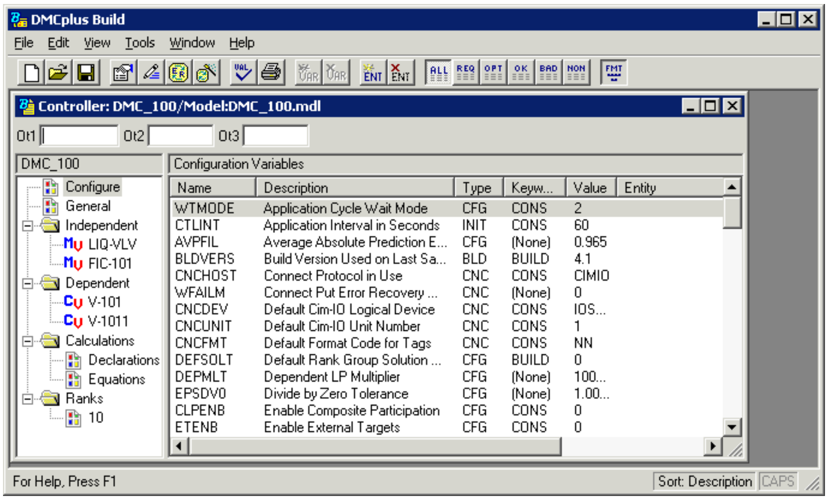
If HYSYS has already generated the MDL file, you can still import the file into the DMCplus Model to review the generated model.

The MDL file determines the size of the control problem and all the dynamic relationships between each independent and dependent variables.

Refer to the DMCplus Desktop manual for details.

7. Start the Aspen Build application and build the DMCplus configuration file (*.**ccf**).

Figure 5.109



The CCF file determines where the input and output parameters for the controller will reside, which optional capabilities will be used, and the values assigned to all of its parameters such as limits, switches, and tuning.

8. Open the **app** folder in the DMCplus Online installation directory.
9. Create a folder with the controller name (for example **DMC_100**) and copy the *.mdl and *.ccf files into the folder.
10. Return to HYSYS program.
11. Click the **Load DMCplus Controller** button. You should now be able to run the simulation using the newly created DMCplus Controller.

For information on setting the low and high variable limits, refer to the **Operation Page** section.

You can set the low and high variable limits for MVs and CVs (similar to setpoint range).

The DMCplus Controller requires that the DMCplus Online and DMCplus Desktop programs are installed. Refer to the Aspen Manufacturing Suite Installation Guide for information on installing DMCplus Online and DMCplus Desktop.

Operation Tab

The Operation Tab contains the following pages:

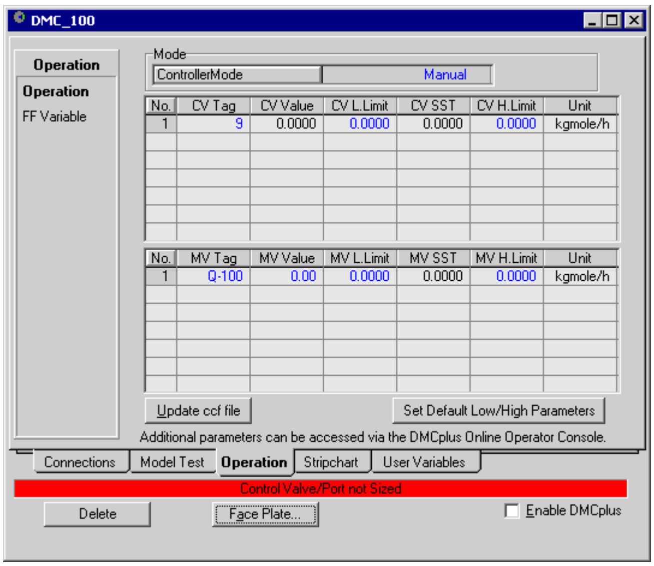
- Operation
- FF Variable

Operation Page

The Operation page allows you to set the controller mode and the low/high limit parameters.

All DMCplus Controllers created can be viewed using the DMCplus GUI. The DMCplus GUI provides more functionality on using the DMCplus Controller.

Figure 5.110



DMCplus uses these low/high parameters to optimize the controlled variable (CV) and manipulated variable (MV) setpoints (steady state target) based on its tuning parameters.

The CV SS target is similar to the MPC Controller setpoint except:

For more information on the MPC Controller, refer to **Section 5.4.5 - MPC Controller**.

- In MPC controller, you specify the CV SS target value.
- In DMCplus controller, you specify the lowest and highest setpoint values, and DMCplus calculates the CV SS target value based on the provided range.

The **Update CCF File** button enables you to export any update configuration results from the **Operation** page into the *.ccf file.

The **Set Default Low / High Parameters** button enables you to populate the CV and MV parameters with default values (refer to the table below):

For CV:	
Low Limit	= current value
High Limit	= current value
For MV:	
Low Limit	= 0
High Limit	= highest value limited by the variable span

The CV and MV SS targets are fixed, if the low/high limit values are the same.

Mode

The DMCplus Controller operates in any of the following modes:

- **Off**. The DMCplus Controller does not manipulate the control valve, although the appropriate information is still tracked.
- **Manual**. Manipulates the DMCplus Controller output manually. In the Manual mode you can enter a value for the Manipulated Variable (MV).
- **Automatic**. The DMCplus Controller reacts to fluctuations in the Controlled Variable (CV) and manipulates the Manipulated Variable (MV) according to the DMCplus algorithm.

Refer to **Section 5.13.2 - Controller Face Plate** for more information.

The mode of the controller may also be set on the Face Plate.

On the Face Plate, you can also view the current value of the CV and MV. You cannot change the low / high limit using the Face Plate.

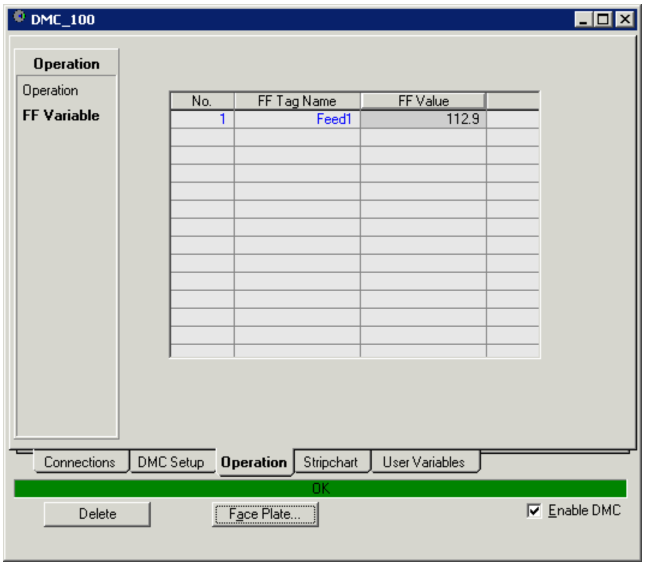
FF Variable Page

Refer to the section on the **Feed Forward (FF)** for more information.

The Feed Forward Variable page allows you to view the controller measured disturbance.

The Feed Forward Variable page will only show the Feed Forward value if the Feed Forward variable has been added on the Connections tab.

Figure 5.111



Stripchart Tab

Refer to **Section 1.3.7 - Stripchart Page / Tab** for more information.

The Stripchart tab allows you to select and create default strip charts containing various variable associated to the operation.

User Variables Tab

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables tab enables you to create and implement your own user variables for the current operation.

5.4.7 Control Valve

The information shown on the Control Valve property view is specific to the associated valve. For instance, the information for a Vapour Valve is different than that for an Energy Stream.

To access the Control Valve property view, click the Control Valve button located at the bottom right corner of the controller operation property view.

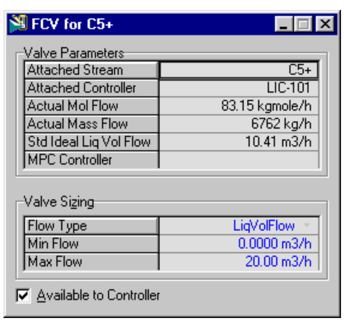
The Control Valve button appears if the OP is a stream.

FCV for a Liquid/Vapour Product Stream from a Vessel

The FCV property view for a material stream consists of two groups:

- Valve Parameters
- Valve Sizing

Figure 5.112



The Valve Parameters group contains flowrate information about the stream with which the Control Valve is associated.

The Valve Sizing group is usually part of the property view that requires specification. This group contains three fields, which are described in the table below:

Field	Description
Flow Type	The type of flow you want to specify: • molar flow • mass flow • liquid volume flow • actual volume flow
Min. Flow	The Minimum flow through the control valve.
Max. Flow	The Maximum flow through the valve.

The Minimum and Maximum flow values define the size of the valve. To simulate a leaky valve, specify a Minimum flow greater than zero. The actual output flow through the Control Valve is calculated using the OP signal (% valve opening):

$$Flow = \frac{OP(\%)}{100}(Maximum - Minimum) + Minimum \tag{5.26}$$

For example, if the Controller OP is 25%, the Control Valve is 25% open, and is passing a flow corresponding to 25% of its operating span. In the case of a liquid valve, if the Minimum and Maximum flow values are 0 and 150 kgmole/h, respectively, the actual flow through the valve is 25% of the range, or 37.5 kgmole/h.

FCV for Energy Stream

The FCV property view that appears is dependent on the type of duty stream selected. There are two types of duty streams:

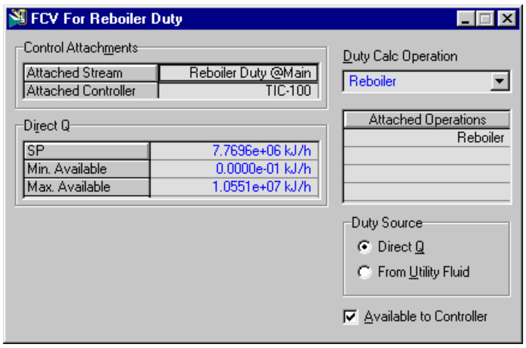
- Direct Q duty consists of a simple power value (in other words, BTU).
- Utility Fluid takes the duty from a utility fluid (in other words, steam) with known properties.

The type of Duty Source specified can be changed at any time by clicking the appropriate radio button in the Duty Source group.

Direct Q Duty Source

This is the Flow Control Valve (FCV) view, when the Duty Source is set to Direct Q in the Duty Source group.

Figure 5.113

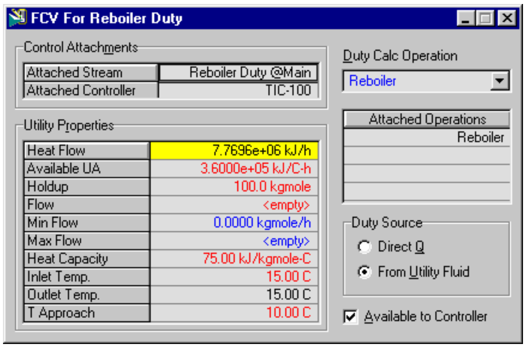


The Attached Stream and Controller appear in the upper left corner of the property view in the Control Attachments group. The specifications required by the property view are all entered into the Direct Q group. In this group, Setpoint (SP) appears, and you may specify the minimum (Min. Available) and maximum (Max. Available) cooling or heating available.

From Utility Fluid Duty Source

As with the Direct Q Duty Source, the attached stream, and controller appear in the upper left corner of the property view.

Figure 5.114



The application of the Utility Fluid information is dependent on the associated operation.

There are several Utility Fluid Parameters, which can be specified in the Utility Properties group:

Parameter	Description
UA	The product of the local overall heat-transfer coefficient and heat-transfer surface area.
Holdup	The total amount of Utility Fluid at any time. The default is 100 kgmole.
Flow	The flowrate of the Utility Fluid.
Min and Max Flow	The minimum and maximum flowrates available for the Utility Fluid.
Heat Capacity	The heat capacity of the Utility Fluid.
Inlet and Outlet Temp	The inlet and outlet temperatures of the Utility Fluid.
T Approach	The operation outlet temperature minus the outlet temperature of the Utility Fluid.

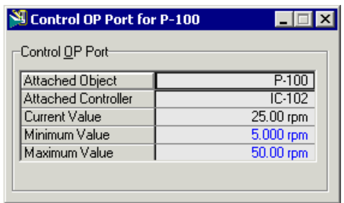
Available to Controller checkbox. When you make the controller connections, and move to the Control Valve property view (by clicking the Control Valve button on the PID Controller property view), the **Available to Controller** checkbox is automatically selected. HYSYS assumes that because you installed a new controller on the valve, you probably want to make it available to the Controller.

5.4.8 Control OP Port

To access the Control OP Port property view, click the **Control OP Port** button at the bottom right corner of the control operation property view.

The Control OP Port button appears when the OP is not a stream and a range of specified values is required.

Figure 5.115



The following table lists and describes the common options in the Control OP Port property view:

Object	Description
Attached Object cell	Displays the name of the output target object attached to the controller.
Attached Controller cell	Displays the name of the controller attached to the output target object.
Current Value cell	Displays the current value of the output target object variable.
Minimum Value cell	Enables you to specify the minimum value for the output target object variable.
Maximum Value cell	Enables you to specify the maximum value for the output target object variable.

5.5 Digital Point

The Digital Point is an On/Off Controller. You specify the Process Variable (PV) you want to monitor, and the output (OP) stream which you are controlling. When the PV reaches a specified threshold value, the Digital Point either turns the OP On or Off, depending on how you have set up the Digital Point.

The PV is optional; if you do not attach a Process Variable Source, the Digital Point operates in Manual mode.

5.5.1 Digital Point Property View

There are two ways that you can add a Digital Point to your simulation:

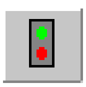
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Logicals** radio button.
3. From the list of available unit operations, select **Digital Pt**.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

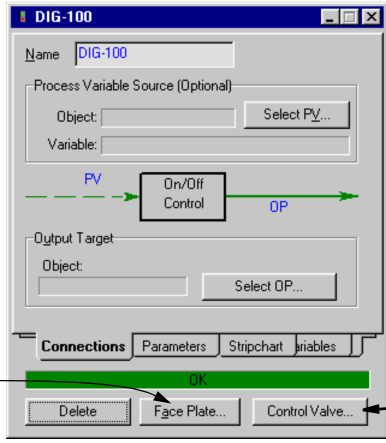
   You can also open the Object Palette by pressing **F4**.
2. Double-click the **Digital Control Point** icon.



Digital Control Point icon

The Digital Point property view appears.

Figure 5.116



Click the **Face Plate** button to view the Controller Face Plate.

Click the **Control Valve** button to view the Control Valve property view.

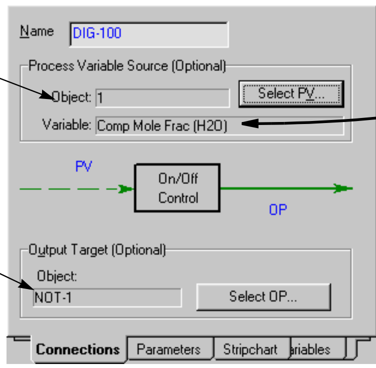
5.5.2 Connections Tab

The Process Variable Source and Output Target are both optional connections. No error is shown when these are not connected nor does an error appear in the Status List Window.

Figure 5.117

The Process Variable object (stream or operation) that owns the variable you want to control.

The Output object is the stream, which is controlled by the Digital Point.



The Process Variable you want to control.

The optional connections feature allows the controller to be in Manual mode, and have its OPState imported into a Spreadsheet and used in further calculations in the model. This configuration can only be used for Manual mode.

To run the controller in Automatic mode, you require a Process Variable Source input. With only the input connected, the Digital Point acts as a digital input indicator. With both the input and output specified the Digital Point can be used to determine its state from its PV and then take a discrete action.

Refer to **Section 1.3.9 - Variable Navigator Property View** for information.

To specify the controller input, use the **Select PV** button to access the Variable Navigator property view, which allows you to simultaneously target the PV Object and Variable. Similarly, use the **Select OP** button to choose the Output Target.

> **The flow of the OP Output is manipulated by the Digital Point Controller.**

5.5.3 Parameters Tab

The Parameters tab provides three different modes of operation:

- Off
- Manual
- Auto

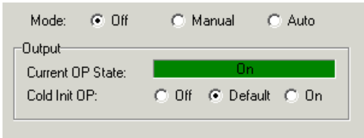
For each of these modes the Parameters tab is made up of a number of groups: Output, Manual/Auto Operational Parameters, and Faceplate PV Configuration.

Off Mode

When Off mode is selected, you cannot adjust the OP State. Notice that if you turn the controller Off while running the simulation, it retains the current OP State (Off or On). Thus, turning the Controller off is not necessarily the same as leaving the Controller out of the simulation.

Only the Output group, displaying the current OP State, is visible when in the Off mode.

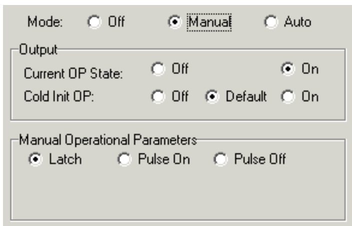
Figure 5.118



Manual Mode

When Manual mode is selected you can adjust the OP State from the Faceplate or this tab. Two groups are visible when in this mode: Output and Manual Operation Parameters.

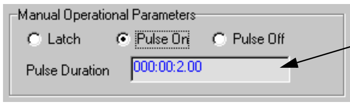
Figure 5.119



The Output group allows you to toggle the OP state on and off. The Operational Parameters group allows you to select one of the three options described in the table below.

Option	Description
Latch	Holds the current OP State to what is specified in the Output group.
Pulse On	Allows the OP State to Pulse On for a specified period and fall back to the Off state.
Pulse Off	Allows the OP State to Pulse Off for a specified period and fall back to the On state.

Figure 5.120

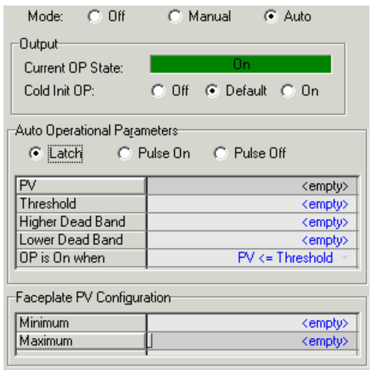


Allows you to specify the duration of the pulse.

Auto Mode

When Auto mode is selected, HYSYS automatically operates the Digital Point, setting the OP State Off or On when required, using the information you provided for the Threshold and OP Status. Three groups are visible when in this mode: Output, Auto Operational Parameters, and Faceplate PV Configuration.

Figure 5.121



Since HYSYS is automatically adjusting the controller, the Output group simply displays the OP State. Like Manual mode, the Auto Operation Parameters group allows you to select one of the three options:

- Latch
- Pulse On
- Pulse Off

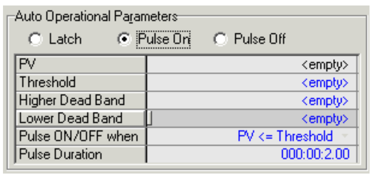
When Latch is selected the following parameters appear.

Parameter	Description
PV	The actual value of the PV (Process Variable).
Threshold	The value of the PV which determines when the controller switches the OP on or off.
Higher Deadband	Allows you to specify the upper deviation of the threshold value.
Lower Deadband	Allows you to specify the lower deviation of the threshold value.
OP On/Off when	Allows you to set the condition when the OP state is on or off.

For more information regarding how Digital Point logical operation determines when to turn the OP state on or off, refer to **Threshold and Dead Band for Latch** section.

For both the Pulse On and Pulse Off options the parameters are the same as the Latch option. However the pulse options both require you to specify a Pulse Duration.

Figure 5.122



For more information regarding how Digital Point logical operation determines state of the Pulse (on or off), refer to **Threshold and Dead Band for Pulse** section.

The Face Plate PV Configuration group allows you to specify the minimum and maximum PV range. This is the range shown on the controllers Face Plate.

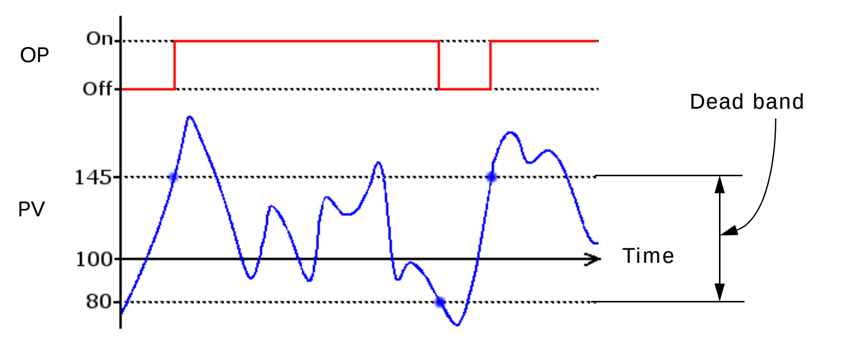
Threshold and Dead Band for Latch

For the Latch option, the OP (output) switches states (on or off) when the PV (Process Variable) value reaches the set point value. The set point value is accompanied with an adjustable differential gap or dead band value, this value allows small deviations to occur in the PV value without triggering changes to the OP state.

The following is an example of how the Latch option operates. Assume you have a Digital Point operation with the following parameters:

Parameter	Value
Threshold	100
Higher dead band	45
Lower dead band	20
OP is on when	PV >= Threshold

Figure 5.123



- Red line indicates the path of the OP vs. time.
- Blue line indicates the path of the PV vs. time.

The following situations illustrates when the OP is turned on or off.

- **Initial State**. If PV value starts at 70 and OP is off, the OP stays off until the PV value rises above or to equal 145, than OP is turned on.
- **Intermediate State**. If PV value rises and falls above value 80 and OP state is already on, then OP remains on.
- **Intermediate State**. If PV value falls below or equal value 80, OP is turned off.

If PV value starts at 150 and OP state is off, then OP remains off until the PV value falls below 80 and rises above 145, after PV reaches or rises above 145 the OP state is turned on.

The above situation is the same for Latch option with OP is on when PV <= Threshold, with the reverse effect. In other words, OP is turned on when PV value passes through the lower dead band value, and OP is turned off when PV value passes through the higher dead band value.

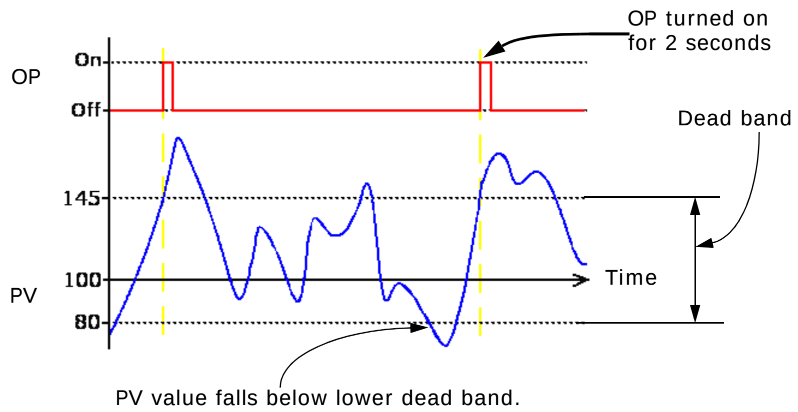
Threshold and Dead Band for Pulse

For the Pulse option, the OP (output) remains in a state (on or off) until the PV (Process Variable) value reaches the set point value, then OP switches state briefly and returns back to its default state (like a pulse). The set point value is accompanied with an adjustable differential gap or dead band value, this value allows small deviations to occur in the PV value without triggering the OP state pulse.

The following is an example of how the Pulse option operates. Assume you have a Digital Point operation with the following parameters:

Parameter	Value
Pulse	Off. The OP default state is off.
Threshold	100
Higher dead band	45
Lower dead band	20
Pulse ON/OFF when	PV >= Threshold
Pulse Duration	2 seconds

Figure 5.124



- Red line indicates the path of the OP vs. time.
- Blue line indicates the path of the PV vs. time.

The following situations illustrates when the OP is turned on or off.

- If PV value starts at 70, the OP is off until the PV value rises to equal 145, than OP is on for 2 seconds.
- After OP state has pulsed on once, OP remains off if PV value rises and falls above the value 80.

If PV value starts at 150, OP is off until the PV value falls below 80 and rises above 145, after PV reaches 145 the OP state is on for 2 seconds.

5.5.4 Stripchart Tab

Refer to **Section 1.3.7 - Stripchart Page / Tab** for more information.

The Stripchart tab allows you to select and create default strip charts containing various variable associated to the operation.

5.5.5 User Variables Tab

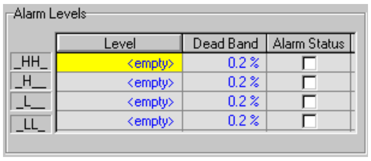
For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables tab enables you to create and implement your own user variables for the current operation.

5.5.6 Alarm Levels Tab

The Alarms tab allows you to set alarm limits for the controller.

Figure 5.125



The Alarm Level group allows you to set and configure the alarm points for a selected signal type. There are four alarm points that can be configured:

- LowLow
- Low
- High
- HighHigh

The alarm points should be specified in the descending order from HighHigh to LowLow points. You cannot specify the value of the Low and LowLow alarm points to be higher than the signal value. Similarly, the High and HighHigh alarm points cannot be specified a value lower than the signal value. Also, no two alarm points can have a similar values. In addition, the user can specify a deadband for a given set of alarms. This can be helpful in situations where the signal is “noisy” to avoid constant triggering of the alarm. If a deadband is specified, you have to specify the alarm points so that their difference is greater than the deadband. At present the range for the allowable deadband is as follows:

$0.0\% \leq \text{deadband} \leq 1.5\%$ of the signal range.

The above limits are set internally and are not available for adjustment by the user!

The Alarm Status displays the recently violated alarm for each alarm point.

5.6 Parametric Unit Operation

The Parametric Unit operation allows selected unit operations, streams, and variables to be solved using a Parametric model. The main function of the Parametric model is to approximate an existing HYSYS model. To build the Parametric model, the Parametric Utility tool is required.

For more information on this utility, refer to **Section 14.14 - Parametric Utility**.

The Parametric utility integrates Neural Network (NN) technology into its framework. A data file with the appropriate data can be used in place of the Parametric Utility.

Using a Parametric model with neural network capability to approximate a HYSYS model significantly improves the robustness of the model, reduces its calculation time, and improves the overall on-line performance. The accuracy of the model depends upon the data available and type of model being approximated.

In the flowsheet, the parametric unit operation essentially “pulls out” a collection of HYSYS unit operations and replaces them. Therefore, this unit operation can be thought of as a “black box” with inputs and outputs. When the flowsheet is solved, the Parametric model is used in place of the individual HYSYS unit operation models.

5.6.1 Parametric Unit Operation Property View

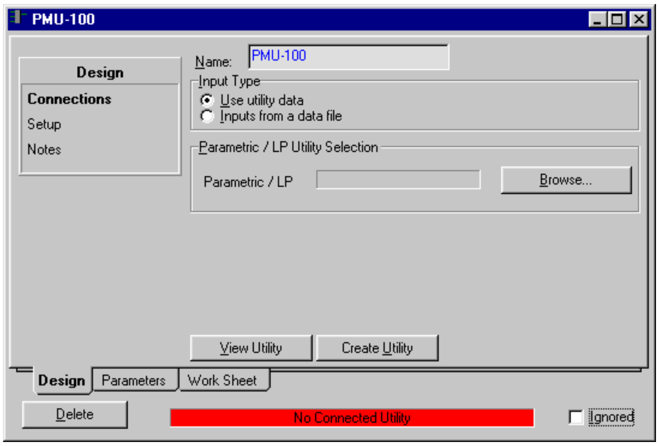
To add a Parametric Unit Operation to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Logicals** radio button.
3. From the list of available unit operations, select **Parametric Unit Operation**.
4. Click the **Add** button.

The Parametric Unit Operation property view appears.

Figure 5.126



5.6.2 Design Tab

The Design tab contains the following pages:

- Connections
- Setup
- Notes

Connections Page

The Connections page allows the parametric unit operation to be connected with the information required for the Parametric model. This information can be found in either a Parametric Utility or a data file. This page always contains a Name field and Input Data group. The rest of the page is different depending on the selected Input Data type.

The Name field allows you to define a unique name for the unit operation. The Input Data group allows you to define where the input data to the Parametric model is to be found. The rest of the property view is altered depending on the radio button selected. The options available are:

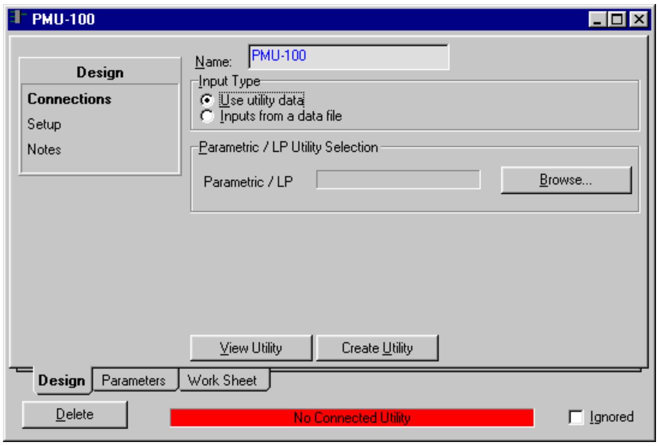
- Use Utility Data
- Inputs from a Data File

Each of the radio buttons are described in the following sections.

Use Utility Data Radio Button

When this radio button is selected the property view appears as shown in the figure below.

Figure 5.127

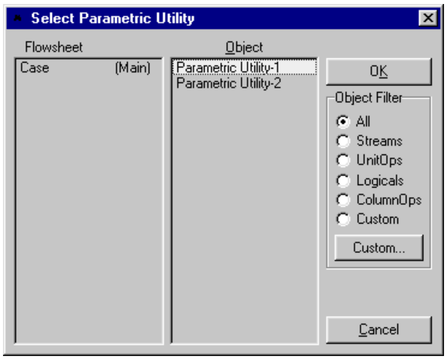


This property view has one additional group, and two additional buttons available.

Parametric/LP Utility Selection Group

If any Parametric Utilities exist in the case, one can be selected as the utility to be used by clicking the **Browse** button. The **Browse** button opens the Select Parametric Utility property view, as shown in the figure below.

Figure 5.128



Select the Parametric Utility to be used for the unit operation, and click the **OK** button.

Create Utility Button

If a Parametric Utility does not exist in the HYSYS case, or you want to create a new Utility, clicking the Create Utility button creates one for use in the Parametric unit operation.

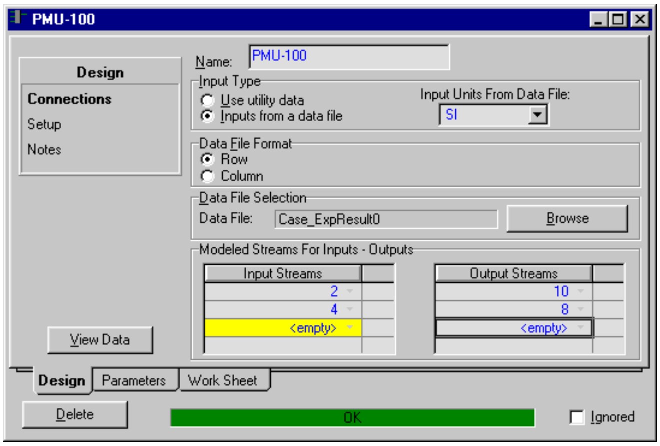
View Utility Button

Clicking the **View Utility** button opens the property view for the selected Parametric Utility.

Inputs from a Data File Radio Button

When using this option, the Parametric unit operation does not have to obtain the model parameter from a utility. Instead, an external data file can be used.

Figure 5.129



Data File Format Group

In the Data File Format group, you can select the format of information to be stored in the *.dat file.

- The format for the Row is:

  $input_{11}$, $input_{21}$, $input_{31}$, ...
  $output_{11}$, $output_{21}$, $output_{31}$, ...
  $input_{12}$, $input_{22}$, $input_{32}$, ...
  $output_{12}$, $output_{22}$, $output_{32}$, ...
- The format for the Column is:

$input_{11}$	$input_{12}$	$output_{11}$	$output_{12}$
$input_{21}$	$output_{22}$	$output_{21}$	$output_{22}$
$input_{31}$	$input_{32}$	$output_{31}$	$output_{32}$
...	...	...	...

Data File Selection Group

Clicking the Browse button allows you to navigate, and locate the data file that contains the required information for the Parametric model. The information in the file is comma delimited, and is stored in a *.dat file.

Input Units from Data File Field

Using the drop-down list, the units used in the data file can be defined.

Modeled Streams for Input–Output Group

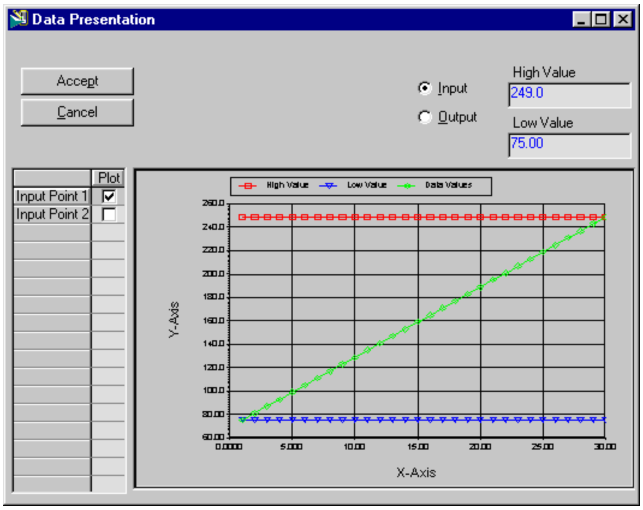
The input and output streams that are being modeled can be selected from the list of existing streams in the drop-down list. A new stream can be created and used in the Parametric unit operation by entering a new stream name in the appropriate cell.

View Data Button

The **View Data** button is available in both the Connections page and Setup page. Clicking the **View Data** button opens the Data Presentation property view.

In this property view, you can see the data from the *.dat file in graph format. The radio button and checkbox in the Plot column allow you to select which data set appears on the graph.

Figure 5.130



You can also specify the high and low limits for each data set, using the High Value and Low Value fields. When you specify the range of data set, the Parametric Unit Operation only takes the data within the range to use for training. So the High Value and Low Value fields allow you to discard any initial and end data values that may be inaccurate.

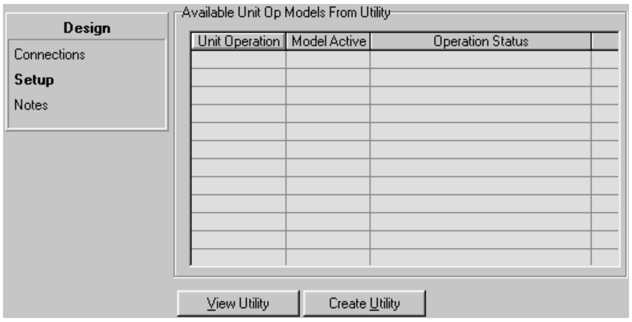
Setup Page

The appearance of this page, like the Connections page, is dependant on the radio button selected in the Input Data group on the Connections page.

Use Utility Data Radio Button

When the Use Utility Data radio button is selected, the property view appears as shown in the figure below.

Figure 5.131



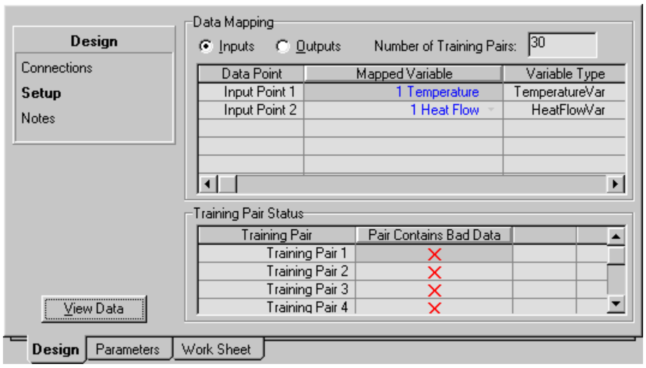
Available Unit Op Models from Utility Group

When the Use Utility Data radio button is selected on the Connections page, only one group is available on the Setup page. In this table, all unit operations that have a Parametric model in the selected Parametric Utility appear. The name of the unit operation, the status of the model activity, and the operation status of the unit operations are all displayed. The Model Activity can be chosen to be active or inactive by clicking the checkbox. When the model is inactive, it is only removed from the Parametric Utility.

Inputs from a Data File Radio Button

When the Inputs from a Data File radio button is selected, the property view appears as shown in the figure below.

Figure 5.132



The Setup page contains two groups:

- Data Mapping
- Training Pair Status

Data Mapping Group

Two radio buttons are available in this group: Inputs and Outputs. The table below describes the properties displayed.

Property	Description
Number of Training Pairs	The number of data sets read in from the data file.
Data Point	A specific data within the data file.
Mapped Variable	The variable in the attached stream that is associated to the data set.
Variable Type	The variable type of the data point, which is selected from the drop-down list.
Identifier	Allows you to enter a unique name to identify the data points.
Low and High Value	The minimum and maximum values in the data set.

Property	Description
Bad Variable Status	If an 'X' appears, the data is good. If a checkmark appears, there is bad data in the data set.
Current Value	The value used in the worksheet after training.

Training Pair Status Group

Displays the individual training pairs, and indicates whether the pair contains bad data. If an 'X' appears, there is no bad data. If a checkmark appears, there is bad data in the data set.

A training pair is defined as a set of input and output data.

Notes Page

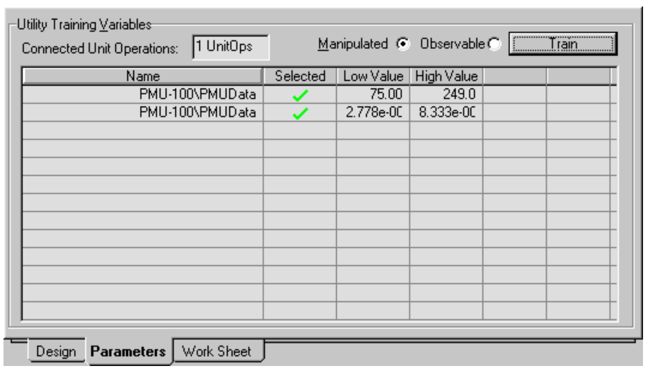
For more information, refer to **Section 1.3.5 - Notes Page/Tab**.

The Notes page provides a text editor, where you can record any comments or information regarding the operation or to your simulation case in general.

5.6.3 Parameters Tab

The Parameters tab displays the training variables of the attached Parametric Utility.

Figure 5.133



There are four main objects in the Parameters tab:

- **Connected Unit Operations**. The number of unit operations connected to the Parametric unit operation appears in this field.
- **Manipulated Variables**. By selecting the Manipulated radio button, the manipulated variables in the Parametric model appear.

  The manipulated variables are the variables being modified in the Parametric Utility and obtained from the HYSYS PFD model simulation. The name of the variable appears, and the selected status is shown. You can select or deselect the variable for use in the parametric model by clicking the checkbox. The lower and upper values used for training are also displayed.
- **Observable Variables**. By selecting the Observable radio button, the observable variables in the Parametric model appear.

  The observable variable is the same as the Observable variable in the Parametric Utility. Observable variables are the HYSYS variables whose values are known and used as training data when calculating the Parametric model. The name of the variable appears and the selected status is shown. You can select or deselect the variable for use in the Parametric model calculation by clicking the checkbox. The lower and upper values for training are also displayed.
- **Train Button**. Clicking the Train button initializes the Parametric Utility training engine to determine the parameters for the Parametric model.

  The Parametric model approximates the HYSYS model in the sense that, given the same values of the training input variables, the values of the output variables of the Parametric model must be close to the values from the HYSYS model.

It is important to realize that there are no methods for training neural networks that can “magically” create information that is not contained in the training data. The neural network model is only as good as its training data.

5.6.4 Worksheet Tab

For more information on the Workbook, refer to **Section 7.23 - Workbook** in the **HYSYS User Guide**.

The Worksheet tab displays the various Conditions, Properties, and Compositions of the unit operations, streams, and variables that are using the Parametric model. From here you can use the neural network instead of the flowsheet, and where the training pairs have been used from a file, see how the neural network has modeled the operation from which your training pairs were generated. These objects appear as different pages on the tab.

5.7 Recycle

The capability of any flowsheet simulator to solve recycles reliably and efficiently is critical. HYSYS has inherent advantages over other simulators in this respect. It has the unique ability to back-calculate through many operations in a non-sequential manner, allowing many problems with recycle loops to be solved explicitly. For example, most heat recycles can be solved explicitly (without a Recycle operation). Material recycles, where downstream material mixes with upstream material, require a Recycle operation.

The Recycle installs a theoretical block in the process stream. The stream conditions can be transferred either in a forward or backward direction between the inlet and outlet streams of this block. In terms of the solution, there are assumed values and calculated values for each of the variables in the inlet and outlet streams. Depending on the direction of transfer, the assumed value can exist in either the inlet or outlet stream. For example, if the user selects Backward for the transfer direction of the Temperature variable, the assumed value is the Inlet stream temperature and the calculated value is the Outlet stream temperature.

The following steps take place during the convergence process:

1. HYSYS uses the assumed values and solves the flowsheet around the recycle.
2. HYSYS then compares the assumed values in the attached streams to the calculated values in the opposite stream.
3. Based on the difference between the assumed and calculated values, HYSYS generates new values to overwrite the previous assumed values.
4. The calculation process repeats until the calculated values match the assumed values within specified tolerances.

5.7.1 Recycle Property View

There are two ways that you can add a Recycle to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.

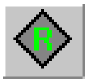
2. Click the **Logicals** radio button.
3. From the list of available unit operations, select **Recycle**.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.

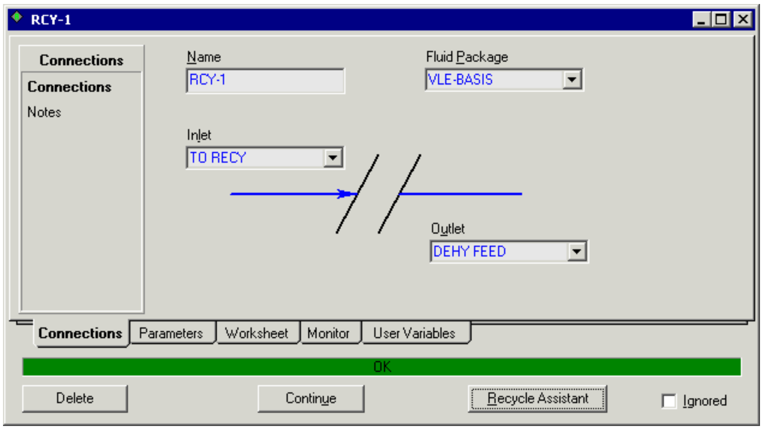
2. Double-click the **Recycle** icon.



Recycle icon

The Recycle property view appears.

Figure 5.134



Object	Description
Continue button	Enables you to run the calculation after the maximum iteration has been reached.
Recycle Assistant button	Enables you to access the Recycle Assistant property view.

Refer to **Section 5.7.9 - Recycle Assistant Property View** for more information.

5.7.2 Connections Tab

The Connections tab contains the following pages:

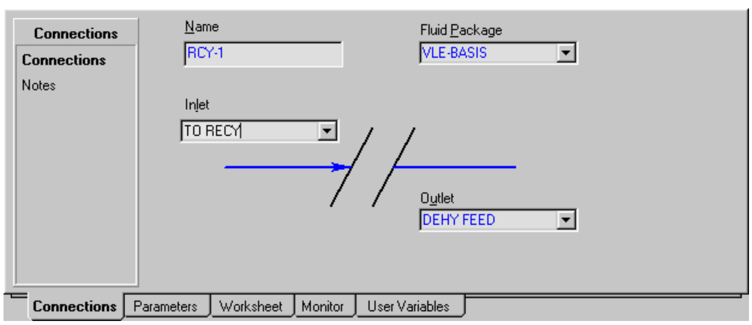
- Connections
- Notes

Connections Page

The Connections page consists of the four fields:

- **Name**. The name of the Recycle operation.
- **Inlet**. Holds the inlet stream, which is the latest calculated recycle; it is always a product stream from a unit operation.
- **Outlet**. Contains the outlet stream, which is the latest assumed recycle; it is always a feed stream to a unit operation.
- **Fluid Package**. The fluid package associated to the operation can be selected by entering the fluid package name or using the drop-down list.

Figure 5.135



Notes Page

For more information, refer to **Section 1.3.5 - Notes Page/Tab**.

The Notes page provides a text editor, where you can record any comments or information regarding the operation or to your simulation case in general.

5.7.3 Parameters Tab

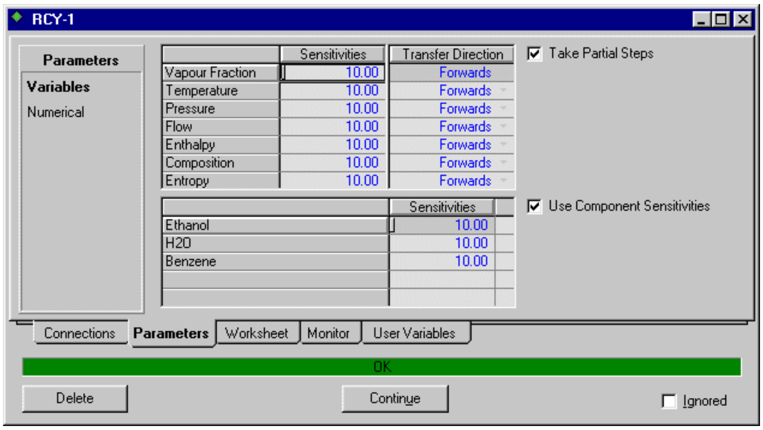
The Parameters tab contains the following pages:

- Variables
- Numerical

Variables Page

HYSYS allows you to set the convergence criteria factor for each of the variables and components listed. The sensitivities values you enter actually serve as a multiplier for HYSYS internal convergence tolerances.

Figure 5.136



The internal absolute tolerances, except flow which is a relative tolerance, are shown in the table below.

HYSYS Internal Tolerances	
Variable	**Internal Tolerance**
Vapour Fraction	0.01
Temperature	0.01
Pressure	0.01
Flow	0.001**
Enthalpy	1.00
Composition	0.0001
Entropy	0.01

**Flow tolerance is relative rather than absolute

The internal Vapour Fraction tolerance, when multiplied by the 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 are 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.

For example, the internal tolerance for temperature is 0.01 and the default multiplier is 10, so the absolute tolerance used by the Recycle convergence algorithm is 0.01*10 = 0.1. Therefore, the assumed temperatures and the calculated temperature must be within 0.1°C (where C is the internal units) of each other if the Recycle is to converge. HYSYS always convert the values entered to the internal units before performing calculations.

A multiplier of 10 (default) is normal, and is recommended for most calculations. Values less than 10 are more stringent; that is, the smaller the multiplier, the tighter the convergence tolerance.

It is not required that each of the multipliers be identical. For example, if you are dealing with ppm levels of crucial components, you can set the Composition tolerance multiplier much tighter (smaller) than the others.

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

- not to transfer
- transfer forward
- transfer backward

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.

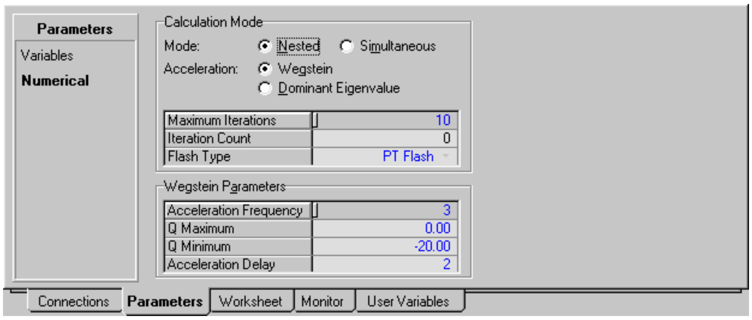
When you select the **Take Partial Steps** checkbox, the Recycle operation takes calculation steps on any variables whenever the calculations are possible. When you clear the checkbox, the Recycle operation waits until all of the inlet stream flowing into the operation is complete before performing the next calculation step. The default setting for this checkbox is clear.

In addition to converging on physical properties basis, the Recycle operation also converges on individual component tolerances. The components in the recycle stream are automatically added to the Recycle logical operation. When you select the **Use Component Sensitivities** checkbox, the single composition sensitivities value is automatically overridden by the individual component sensitivities values and the Recycle operation takes calculation steps on each value when applicable. The default setting for this checkbox is inactive. Once you select the **Use Component Sensitivities** checkbox, the tolerances for each component in the recycle stream are listed in the table. By default, the sensitivities value is set at 10.00 for each component. Any changes that you make to the sensitivities value are automatically saved.

Numerical Page

The Numerical page contains the options related to the Wegstein Acceleration Method.

Figure 5.137



This method is used by the Recycle to modify the values it passes from the inlet to outlet streams, rather than using direct substitution.

The table below describes the parameters on the Numerical page.

Refer to the **Type of Recycle** section for more information.

Numerical Parameters	Description
Mode	You can choose between Nested or Simultaneous mode by selecting the respective radio button. The default mode is Nested.
Acceleration	You can choose between two methods of acceleration: • **Wegstein**. Ignores interactions between variables being accelerated. • **Dominant Eigenvalue**. Includes interactions between variables being accelerated. Further, the Dominant Eigenvalue option is superior when dealing with non-ideal systems or systems with strong interactions between components.
Maximum Iterations	The number of iterations before HYSYS stops (the default is 10). You can continue with the calculations by clicking the **Continue** button at the bottom of the Recycle property view.
Iteration Count	The number of iterations before an acceleration step is applied to the next iteration (default is 0).
Flash Type	The Flash method to be implemented by the Recycle unit op.

Numerical Parameters	Description
Acceleration Frequency	The value in this field is the number of steps to go before putting in acceleration. The lower the value, the more often variables get accelerated.
Q maximum / Q minimum	Damping factors for the acceleration step (defaults are 0 and -20).
Acceleration Delay	This delays the acceleration until the specified step (default is 2).

Type of Recycle

There are two choices for the type of Recycle:

- Nested
- Simultaneous

The Nested option results in the Recycle being called whenever it is encountered during the calculations. In contrast, the Simultaneous option causes all Recycles to be invoked at the same time once all recycle streams have been calculated. If your flowsheet has a single Recycle operation, or if you have multiple recycles which are not connected, use the Nested option (default). If your flowsheet has multiple inter-connected recycles, use the Simultaneous type.

The Calculation Level for a Recycle (accessed under Main Properties) is 3500, compared to 500 for most streams and operations. This means that the Recycle is solved last among unknown operations. You can set the relative solving order of Recycles by modifying the Calculation Level.

There are several additional points worth noting about the Recycle:

- When the Recycle cannot be solved in the number of iterations you specify, HYSYS stops. If you decide that the problem may converge with more iterations, simply click the Continue button. The Recycle initializes the iteration counter and continues until a solution is found or it again runs out of iterations.
- If your problem does not converge in a reasonable number of iterations, there are probably constraints in your flowsheet which make it impossible to solve. In particular, if the size of the recycle stream keeps growing, it is likely that the flowsheet does not permit all

of the material entering the flowsheet to leave. An example of this occurs in gas plants when you are trying to make a liquid product with a low vapour pressure and a vapour product which must remain free of liquids even at cold temperatures. Often, this leaves no place for intermediate components like propane and butane to go, so they accumulate in the plant recycle streams. It is also possible that the tolerance is too tight for one or more of the Recycle variables and cannot be satisfied. This can readily be determined by examining the convergence history, and comparing the unconverged variable deviations with their tolerances.

The Monitor tab provides a history of the Recycle calculations.

- The logical operations (such as the Recycle, Adjust and Controller) are different from other operations in that they actually modify the specifications of a stream. As a result, if you remove any of these operations, the outlet stream specifications remain. Thus, nothing in the flowsheet is “forgotten” for these operations. You can Delete or Ignore a Recycle when you want to make flowsheet modifications, but do not want to invoke the iterative routines.
- Tolerance settings are important to a successful Recycle solution. This is especially true when multiple recycles are involved. If there is no interaction among the recycles, or if they are inter-connected and are being solved simultaneously, tolerance values can be identical for all Recycles if desired. However, if the Recycles are nested, tolerances should be made increasingly tighter as you go from the outermost to the innermost Recycle. Without this precaution, the outside Recycle may not converge.

Maximum Number of Iterations

When HYSYS has reached the maximum number of iterations, a warning message appears stating that the Recycle failed to converge in the specified number of iterations. You can then choose whether or not to continue calculations.

If you are starting a new flowsheet, use a small number of Maximum Iterations, such as 3. Once it is evident that the calculations are proceeding well, the count can be increased. The iterations required depend not only on the complexity of your flowsheet, but also on your initial estimate and the convergence tolerances you use.

Damping Factors - Qmax and Qmin

The Wegstein acceleration method uses the results of previous iterations in making its guesses for the recycle stream variables. Assumed values are calculated as follows:

$$X_{n+1} = QX_n + (1 - Q)Y_n \tag{5.27}$$

where:

X = assumed value

Y = calculated value

n = iteration number

Q = acceleration factor

HYSYS determines the actual acceleration (Q) to apply based on the amount of change between successive iterations. The values for Q_{max} and Q_{min} set bounds on the amount of acceleration applied. Note from the equation that when $Q = 0$, direct replacement is used. When Q is negative, acceleration is used. When Q is positive and smaller than 1, damping occurs.

If you are finding that your Recycle is still oscillating, even with the Iteration Count set to ensure direct replacement, you can input a slightly larger value for Qmax to damp the direct replacement.

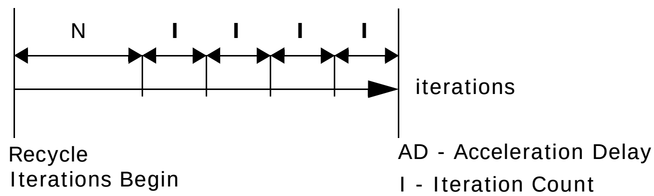
Iteration Count

The Iteration Count is the number of Recycle iterations before an acceleration step is applied when calculating the next assumed recycle value. The default count is 3; after three iterations (assuming the Acceleration Delay is less than 3), the assumed and calculated recycle values are compared and the Wegstein acceleration factor is determined and applied to the next assumed value. When the acceleration factor is not being used (in all iterations up to the Iteration Count), the next assumed value is determined by direct replacement.

Notice that Acceleration Delay takes precedence over the Iteration Count. This means that for an Acceleration Delay value of *x*, the initial *x* iterations use direct replacement, even if the Iteration Count is set to less than *x*. The *x+1* iteration uses the acceleration after which the Iteration Count applies.

Figure 5.138

if $I \leq AD$ then $N = AD+1$ else if $I > AD$ then $N = I$



Although acceleration generally works well for most problems, in some cases it may result in over-correction, oscillation, and possibly non-convergence. Examples of this type of problem include highly-sensitive recycles and multiple recycle problems with strong interactions among recycles. In cases such as these, direct replacement may be the best method for all iterations. To eliminate the use of acceleration, simply set the Iteration Count (or Acceleration Delay) to a very high number of iterations (for example, 100) which is never reached. In the rare instance where even direct replacement causes excessive over-corrections, damping is required. Use the set of parameters discussed below to control this.

Acceleration Delay

The Acceleration Delay parameter delays the acceleration until the specified step. This delay applies to the initial set of iterations and once the specified step is reached the Iteration Count is applied. That is to say no acceleration is performed until the delay value is reached and after that iteration the acceleration is applied according to the Iteration count. The default is specified as 2 but now it can be specified to any value. For example, if the 'delay' is set to 5 and the Iteration Count is 3 then the first 5 iterations use direct replacement and the sixth uses acceleration then after every third iteration the acceleration step is applied.

5.7.4 Worksheet Tab

For more information refer to **Section 1.3.10 - Worksheet Tab**.

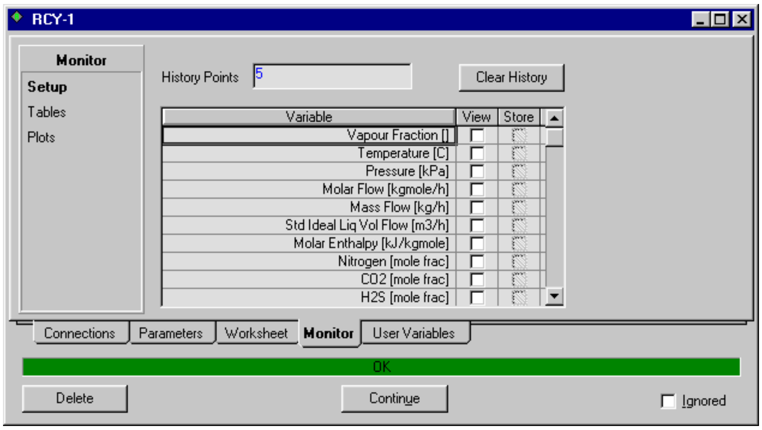
The Worksheet tab displays the various Conditions, Properties, and Compositions of the Feed and Product streams.

5.7.5 Monitor Tab

The Monitor tab contains the following pages:

- Setup
- Tables
- Plots

Figure 5.139

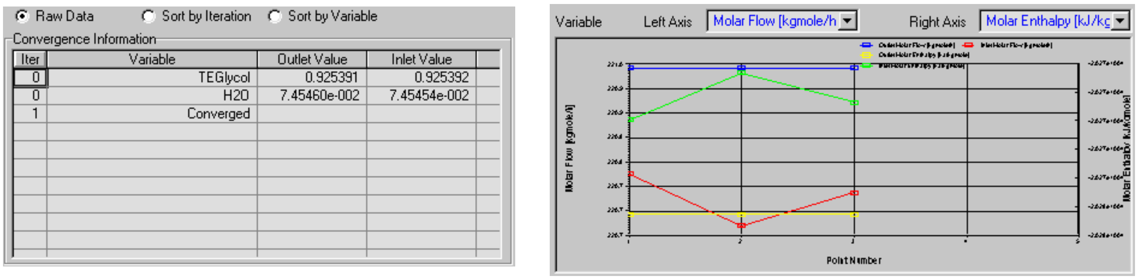


The Setup page allows you to specify which variables you want to view or monitor. To view a variable, select the **View** checkbox corresponding to the variable of interest.

The Tables page and Plots page display the convergence information as the calculations are performed in tabular and graphical format respectively. The inlet value, outlet value, and variable are shown, along with the iteration number.

This is illustrated in the following figure.

Figure 5.140



Tables page

Plots page

5.7.6 User Variables Tab

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables tab enables you to create and implement your own user variables for the current operation.

5.7.7 Calculations

HYSYS provides a very simple means of solving recycle problems, and its interactive nature provides a high degree of control and feedback to the user as to how the solution is proceeding.

In Dynamic mode, HYSYS ignores the Recycle operation. So in the case of the outlet stream, it is identical to the inlet stream.

The Recycle can be set up as a single unit operation to represent a single recycle stream in a process flowsheet, or a number of them can be installed to represent multiple recycles, interconnected or nested, as well as a combination of interconnected and nested recycle loops. Similar to the multi-Adjust operation, the Recycle solves all the recycle loops simultaneously, if requested to do so.

The step-by-step procedure for setting up a recycle is as follows:

1. Make a guess for the assumed values in the stream attached to the recycle operation (temperature, pressure, flow rate, composition). The flow rate can generally be zero, but, obviously, better estimates results in faster convergence.

If the recycle is a feed to a tower, a reasonable estimate is needed to ensure that the column converges the first time it is run.

2. Build your flowsheet until the calculated values in the connected streams can be determined by HYSYS.

The outlet and inlet recycle streams must have different names.

3. Install the Recycle block.

5.7.8 Reducing Convergence Time

Selection of the recycle tear location is vitally important in determining the computer run time to converge the Recycle. Although the physical recycle stream itself is often selected as the tear stream, the flowsheet can be broken at virtually any location. In simulating a complex system, a number of factors must be considered. The following are some general guidelines:

- Choose a Tear Location to Minimize the Number of Recycles

  Reducing the number of locations where the iterative process is required save on total convergence time. Choosing the location of the Recycle depends on the flowsheet topology. Attempt to choose a point such that specifying the assumed values defines as many streams downstream as possible. It generally occurs downstream of gathering points and upstream of distribution points.

Examples include downstream of mixers (often mixing points where the physical recycle combines with the main stream), and upstream of tees, separators, and columns.

- Choose a Tear Location to Minimize the Number of Recycle Variables

  Variables include vapour fraction, temperature, pressure, flow, enthalpy, and composition. Choose the tear stream so that as many variables as possible are fixed, thus effectively eliminating them as variables and increasing convergence stability. Good choices for these locations are at separator inlets, compressor aftercooler outlets, and trim heater outlets.

Avoid choosing tear streams which have variables determined by an Adjust operation.

- Choose a Stable Tear Location

  The tear location can be chosen such that fluctuations in the recycle stream have a minimal effect. For example, by placing the tear in a main stream, instead of the physical recycle, the effect of fluctuations are reduced. The importance of this factor depends on the convergence algorithm. It is more significant when successive substitution is used. Choosing stable tear locations is also important when using simultaneous solution of multi-recycle problems.

5.7.9 Recycle Assistant Property View

The Recycle Assistant property view enables you to find places to insert Recycle unit operation that make the simulation case convergent easily.

The feature is only available in Steady State mode.

There are two other functionalities in the Recycle Assistant feature:

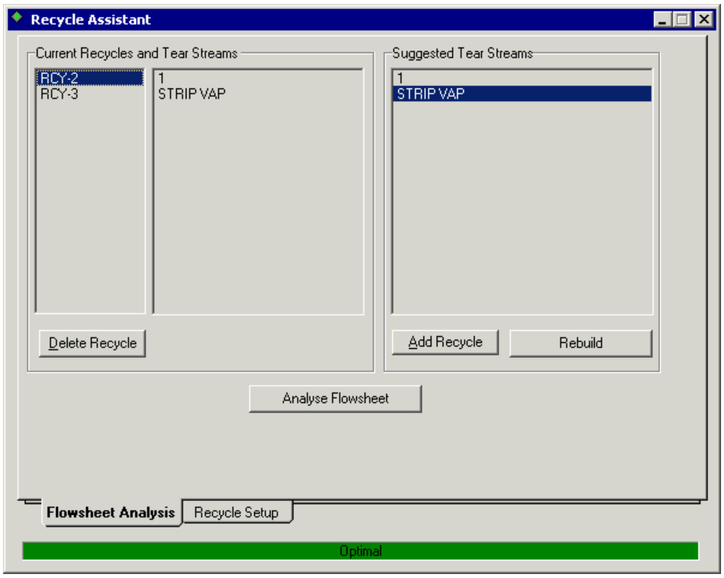
- Enables you to analyse the flowsheet to get suggested tear streams.
- Enables you to delete and add Recycle Unit Op in the Recycle Assistant's interface. For the delete option, only one Recycle Unit op can be deleted at one time.

To access the Recycle Assistant property view, do one of the following:

- In the HYSYS menu bar, select the **Tools | Recycle Assistant** command.
- Open the Recycle operation property view, and click the **Recycle Assistant** button.

The Recycle Assistant property view appears.

Figure 5.141



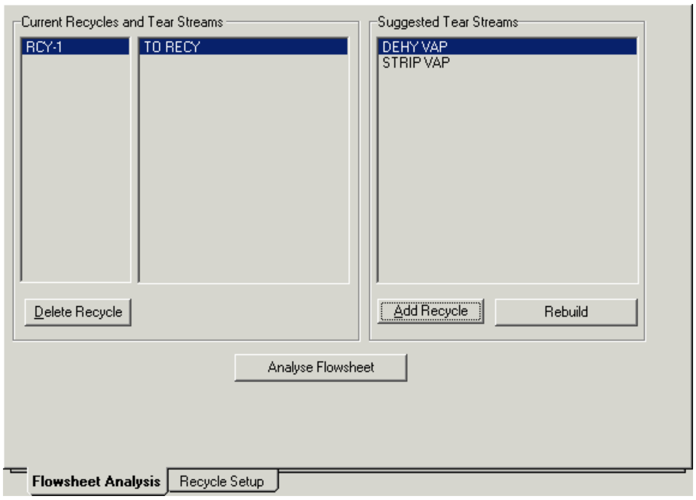
The options in the Recycle Assistant property view are split into the following tabs:

- Flowsheet Analysis
- Recycle Setup

Flowsheet Analysis Tab

The Flowsheet Analysis tab enables you to add, modify, and delete recycle operations and analyse the flowsheet.

Figure 5.142

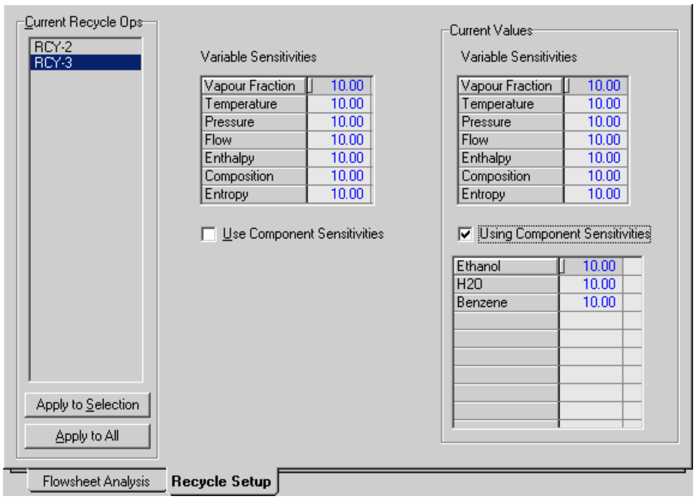


Object	Description
Current Recycles and Tear Streams group	Displays two lists of all the recycles and associate tear streams available in the PFD.
Delete Recycle button	Enables you to delete the selected recycle operation in the list.
Suggested Tear Streams group	Displays a list of possible tear streams available in the PFD.
Add Recycle button	Enables you to add a recycle to the selected tear stream in the list.
Rebuild button	Enables you to modify and optimize the process flow diagram using the suggested tear streams and recycles.
Analyse Flowsheet button	Enables you to update the list of available tear stream in the PFD.

Recycle Setup Tab

The Recycle Setup tab enables you to modify the variable sensitivity of the selected recycle operation.

Figure 5.143



Object	Description
Current Recycle Ops list	Displays the list of available recycle operations in the PFD.
Vapour Fraction cell	Enables you to modify the vapour fraction sensitivity.
Temperature cell	Enables you to modify the temperature sensitivity.
Pressure cell	Enables you to modify the pressure sensitivity.
Flow cell	Enables you to modify the flow rate sensitivity.
Enthalpy cell	Enables you to modify the enthalpy sensitivity.
Composition cell	Enables you to modify the composition sensitivity.
Entropy cell	Enables you to modify the entropy sensitivity.
Using Component Sensitivities checkbox	Enables you to toggle between including or discarding the sensitivity values based on the components in the PFD.
Component table	Enables you to modify the component sensitivity values for all the components in the PFD.
Current Values group	Displays the same options available in the Variable Sensitivities table, except the variable values are taken from the selected recycle operation in the Current Recycle Ops list.

Object	Description
Apply to Selection button	Enables you to apply the modified variable sensitivities values to the selected recycle operations in the Current Recycle Ops list.
Apply to All button	Enables you to apply the modified variable sensitivities values to all the recycle operations in the PFD.

5.8 Selector Block

The Selector Block is a multiple-input single-output controller, that provides signal conditioning capabilities. It determines an *Output value* based on a user-set Input function. For instance, if you want the maximum value of a specific variable for several Input streams to dictate the Output, you would use the Selector Block. A simple example would be where a Selector Control chooses the average temperature from several temperature transmitters in a Column, so that the Reboiler duty can be controlled based on this average.

5.8.1 Selector Block Property View

There are two ways that you can add a Selector Block to your simulation:

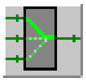
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Logicals** radio button.
3. From the list of available unit operations, select **Selector Block**.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.

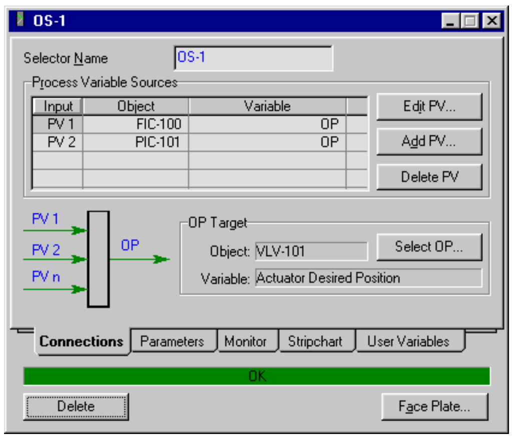


Selector Block icon

2. Double-click the **Selector Block** icon.

The Selector Block property view appears.

Figure 5.144



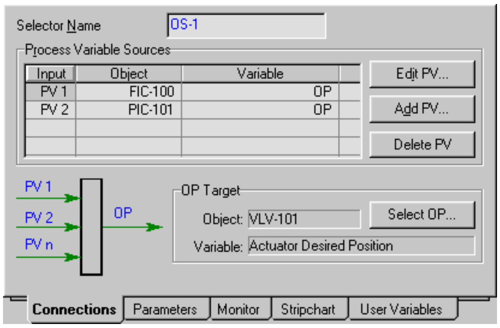
5.8.2 Connections Tab

The Connections tab consists of three objects described in the table below:

Objects	Description
Selector Name	Contains the name of the Selector Block, which can be edited at any time.
Process Variable Sources	Contains the variables the Selector Block considers and inserted into the input function. The Process Variables for the various inputs are targeted by clicking Add PV button, accessing the Variable Navigator. You can edit or delete any current PV by positioning the cursor in the appropriate row, and clicking the **Edit PV** or **Delete PV** buttons. If you add a Variable whose type is inconsistent with the current Input Variables, HYSYS displays an error message. However, you are allowed to retain that Variable.
OP Target	Contains the process variable, which is manipulated by the Selector Block. To select the OP Target click the Select OP button. This button also accesses the Variable Navigator. Notice that it is not necessary for the Target Variable type to match the Input Variable type.

Refer to **Section 1.3.9 - Variable Navigator Property View** for information.

Figure 5.145



5.8.3 Parameters Tab

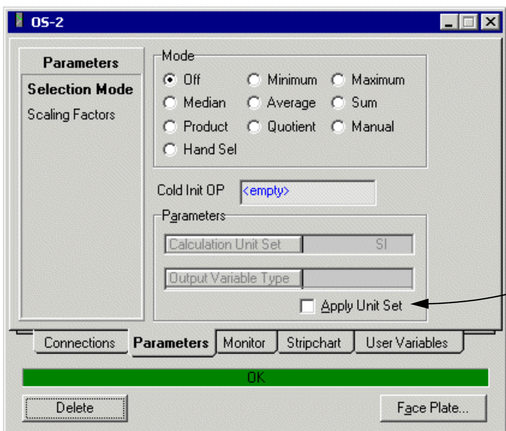
The Parameters tab contains the following pages:

- Selection Mode
- Scaling Factor

Selection Mode Page

The Selection Mode page allows you to select the mode and parameters of the operation.

Figure 5.146



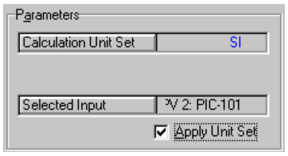
When the **Apply Unit Set** checkbox is clear, the operation uses default internal HYSYS units (in other words, no real unit conversion) for the calculations.

In the Mode group, you can choose from the following modes:

Modes	Description
Off	Select this mode to disable the Selector Block. The function of this mode is similar to the Ignored checkbox in the unit operations.
Minimum	The minimum value from the list of Input Variables is passed to the Output stream.
Maximum	The maximum value from the list of Input Variables is passed to the Output stream.
Median	The median value of the Input Variables is passed to the Output stream. If there are an even number of Input Variables, then the higher of the two middle values is passed to the Output stream.
Average	The average of the Input Variables is passed to the Output stream.
Sum	The sum of the Input Variables is passed to the Output stream.
Product	The product of the Input Variables is passed to the Output stream.
Quotient	The quotient of the Input Variables is passed to the Output stream.
Manual	This mode is similar to Manual mode in the PID controller. This mode allows you to specify the OP directly.
Hand Sel	Allows you to select which Input Variable value is written to the OP. You can select the PV value using the Selected Input field.

When the **Apply Unit Set** checkbox is selected, you can select a unit set and variable type for the calculations and output respectively.

Figure 5.147



In the Parameters group, you can specify the following parameters:

Refer to **Section 12.3.1 - Units Page** in the **HYSYS User Guide** for more information.

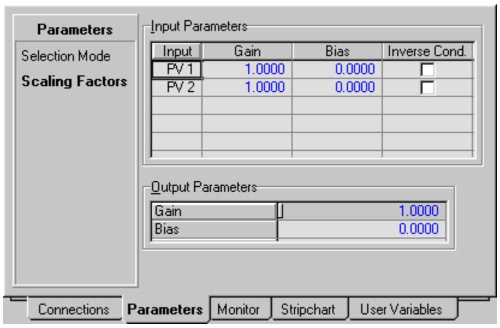
- **Calculation Unit Set**. You can select the unit set you want the calculations done with using the drop-down list. There are three standard selections: SI, EuroSI, and Field. You can create your own unit set in the Session Preferences property view.

- **Selected Input**. You can select the PV source using the drop-down list in this field. This field is only available for the Minimum, Maximum, and Hand Sel modes.

Scaling Factors Page

The Scaling Factors page allows you to manipulate the input and output parameters.

Figure 5.148



The Output is a function of the Mode, Gain, and Bias, where the Input function is dependent on the mode:

$$\text{Output} = f(\text{Inputs}) \times \text{Gain} + \text{Bias} \quad (5.28)$$

The Input function is multiplied by the Gain. In effect, the gain tells how much the output variable changes per unit change in the input function. The Bias is added to the product of the Input function and Gain.

> **If you want to view the Input function without any Gain or Bias adjustment, set the Gain to one and the Bias to zero.**

Inputs can be individually scaled before the output calculations.

$$\text{Input}_{scaled} = \text{Input} \times \text{Gain} + \text{Bias} \quad (5.29)$$

The Inverse Cond. checkbox is used when the selector is writing back to its PV, and you are required to scale the input value backwards.

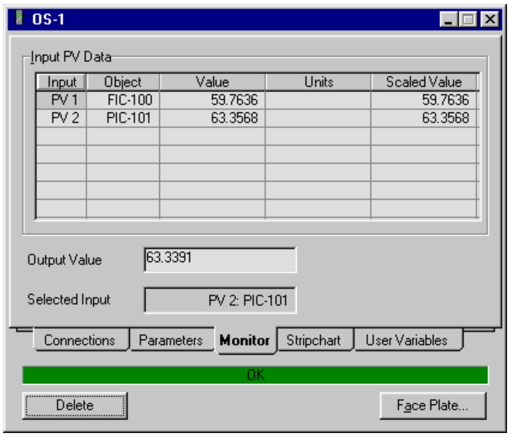
$$\text{Input} = \frac{\text{Input}_{scaled} - \text{Bias}}{\text{Gain}} \tag{5.30}$$

5.8.4 Monitor Tab

The Monitor tab displays the results of the Selector Block. It consists of two objects:

- **Input PV Data**. This group contains the current values of the input process variable.
- **Output Value**. This display field contains the current value of the PV for each of the input variables and the Output Value is also displayed on the Parameters tab.

Figure 5.149



The Output value is not displayed until the Integrator has been started.

Example

In the simple example shown here, the median value of Stream 1, Stream 2, and Stream 3 is passed to the Output, after a Gain of 2 and a Bias of 5 have been applied.

These are the steps:

1. Determine *f(Inputs)*, which in this case is the median of the Input Variables. The median (middle value) temperature of the three streams (10°C, 15°C, 20°C) is 15°C.
2. Determine the Output Value, from the Gain and Bias. The Gain is 2, and the Bias is 5°C.
3. The Output is calculated as follows:

$$\begin{aligned} Output &= f(Inputs) \times Gain + Bias \\ Output &= 15^\circ C \times 2.000 + 5.000^\circ C \\ Output &= 35^\circ C \end{aligned} \tag{5.31}$$

5.8.5 Stripchart Tab

Refer to **Section 1.3.7 - Stripchart Page / Tab** for more information.

The Stripchart tab allows you to select and create default strip charts containing various variable associated to the operation.

5.8.6 User Variables Tab

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables tab enables you to create and implement your own user variables for the current operation.

5.9 Set

The Set is an operation used to set the value of a specific Process Variable (PV) in relation to another PV. The relationship is between the same PV in two like objects; for instance, the temperature of two streams, or the UA of two exchangers.

The Set unit operation can be used in both Dynamic and Steady State mode.

The dependent, or target, variable is defined in terms of the independent, or source, variable according to the following linear relation:

$$Y = MX + B \tag{5.32}$$

where:

Y = dependent (target) variable

X = independent (source) variable

M = multiplier (slope)

B = offset (intercept)

5.9.1 Set Property View

There are two ways that you can add a Set to your simulation:

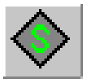
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Logicals** radio button.
3. From the list of available unit operations, select **Set**.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.

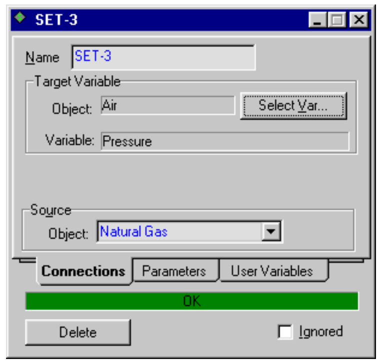


Set icon

2. Double-click the **Set** icon.

The Set property view appears.

Figure 5.150



5.9.2 Connections Tab

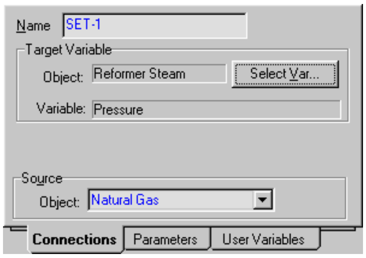
On the Connections tab, you can specify the following information:

Refer to **Section 1.3.9 - Variable Navigator Property View** for more information.

- **Target Object**. The stream or operation to which the dependent variable belongs. This is chosen by clicking the Select Var button. This brings up the Variable Navigator property view.
- **Target Variable**. The type of variable you want to set, for example, temperature, pressure, and flow. The available choices for Variable are dependent on the Object type (stream, heat exchanger, and so forth) Your choice of Variable is automatically assigned to both the Target and Source object.
- **Source Object**. The stream or operation to which the independent variable belongs.

Notice that when you choose an object for the Target, the available objects for the Source are restricted to those of the same object type. For example, if you choose a stream as the Target, only streams are available for the Source.

Figure 5.151

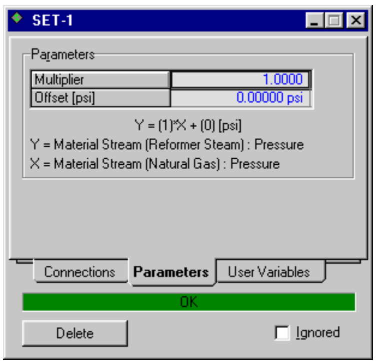


HYSYS solves for either the Source or Target variable, depending on which is known first (bi-directional solution capabilities).

5.9.3 Parameters Tab

On the Parameters tab, you can specify values for the slope (Multiplier) and the intercept (Offset). The default values for the Multiplier and Offset are 1 and 0, respectively.

Figure 5.152



5.9.4 User Variables Tab

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables tab enables you to create and implement your own user variables for the current operation.

5.10 Spreadsheet

The Spreadsheet applies the functionality of Spreadsheet programs to flowsheet modeling. With essentially complete access to all process variables, the Spreadsheet is extremely powerful and has many applications in HYSYS.

The HYSYS Spreadsheet has standard row / column functionality. You can import a variable, or enter a number or formula anywhere in the spreadsheet.

The Spreadsheet can be used to manipulate or perform custom calculations on flowsheet variables. Because it is an operation, calculations are performed automatically; Spreadsheet cells are updated when flowsheet variables change. In Dynamics mode, the Spreadsheet cells are updated when the integrator is running.

One application of the Spreadsheet is the calculation of pressure drop during dynamic operation of a Heat Exchanger. In the HYSYS Heat Exchanger, the pressure drop remains constant on both sides regardless of flow. However, using the Spreadsheet, the actual pressure drop on one or both sides of the exchanger could be calculated as a function of flow.

Complex mathematical formulas can be created, using syntax which is similar to conventional Spreadsheets. Arithmetic, logarithmic, and trigonometric functions are examples of the mathematical functionality available in the Spreadsheet. The Spreadsheet also provides logical programming in addition to its comprehensive mathematical capabilities. Boolean logic is supported, which allows you to compare the value of two or more variables using logical operators, and then perform the appropriate action depending on that result.

You can import virtually any variable in the simulation into the Spreadsheet, and you can export a cell's value to any specifiable field in your simulation. There are two methods of importing and exporting variables to and from the Spreadsheet:

Refer to **Section 1.3.9 - Variable Navigator Property View** for more information.

Methods	Description
Using the Variable Navigator	Do one of the following: • On the Connections tab, click the Add Import or Add Export button. • On the Spreadsheet tab, right-click the cell you want and select export/import command from the object inspection menu. Then using the Variable Navigator, select the variable you want to import or export.
Dragging Variables	Simply right-click the variable value you want to import, and drag it to the desired location in the Spreadsheet. If you are exporting the variable, drag it from the Spreadsheet to an appropriate location. When using the Dragging Variables method, the property views have to be non-modal.

When you are using the Spreadsheet to return a result back to the flowsheet, you must consider its application in terms of the overall calculation sequence, particularly when Recycles are involved.

Refer to **Section 7.2 - Main Properties** in the **HYSYS User Guide** for more information on the Calculation Sequencing option.

If the Spreadsheet performs a calculation and sends the results back upstream, the potential exists for creating inconsistencies as the full effect of the previous Recycle loop has not propagated through the flowsheet. By using the Calculation Sequencing option, you can minimize the potential for problems of this nature.

5.10.1 Spreadsheet Property View

There are two ways that you can add a Spreadsheet to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

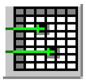
   You can also access the UnitOps property view by pressing **F12**.

2. Click the **Logicals** radio button.
3. From the list of available unit operations, select **Spreadsheet**.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.

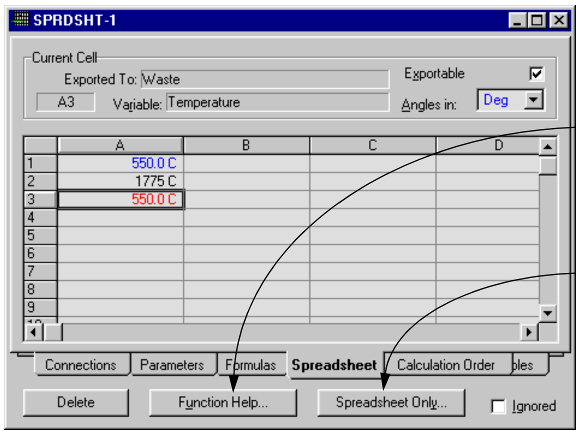


Spreadsheet icon

2. Double-click the **Spreadsheet** icon.

The Spreadsheet property view appears.

Figure 5.153



Click the **Function Help** button to access the list of available spreadsheet functions in HYSYS.

Clic the **Spreadsheet Only** button to open the a property view containing only the spreadsheet and its content.

5.10.2 Spreadsheet Functions

The HYSYS Spreadsheet has extensive mathematical and logical function capability. To view the available Spreadsheet Functions and Expressions, click the **Function Help** button to open the Available Expressions and Functions property view.

All functions must be preceded by "+" (straight math) or "@" (special functions like logarithmic, trigonometric, logical, and so forth).

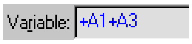
Examples are "+A4/B5" and "@ABS(A4-B5)".

The Available Expressions and Functions property view contains the following tabs:

- Mathematical Expressions
- Logical Expressions
- Mathematical Functions

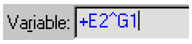
General Math Functions

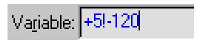
The following arithmetic functions are supported:

General Operations	Method of Application	View
Addition	Use the "+" symbol.	
Subtraction	Use the "-" symbol.	
Multiplication	Use the "*" symbol.	
Division	Use the "/" symbol, typically located on the numeric keypad, or next to the right **SHIFT** key. (Do not use the "\" symbol).	
Absolute Value	"@Abs".	

Several other mathematical functions are also available:

Refer to the **Calculation Hierarchy** section for more information.

Advanced Operations	Method of Application	View
Power	Use the "^" symbol. Example: +3^3 = 27 Example 2: +27^(1/3)=3 Notice that the parentheses are required in this case, since the cube root of 27 (or 27 to the power of one-third) is desired.	
Square Root	"@SQRT". Example: @sqrt(16) = 4 Notice that capitalization is irrelevant. You can also use "@RT" to calculate a square root. (Example: @rt(16)=4)	

Advanced Operations	Method of Application	View
Pi	Simply enter "+pi" to represent the number 3.1415...	
Factorial	Use the “!” symbol. Example: +5!-120=0	

Calculation Hierarchy

The usual hierarchy of calculation is used (Brackets, Exponents, Division and Multiplication, Addition and Subtraction). For example:

```
+6+4/2 = 8 (not 5)
```

since division is performed before addition. However,

```
+(6+4)/2 = 5
```

because any expressions in parentheses are calculated first.

Logarithmic Functions

Log Function	Method of Application	View
Natural Log	“@ln”. Example: @ln(2.73)=1.004	
Base 10 Log	"@log". Example: @log(1000)=3	
Exponential	“@exp”. Example: @exp(3)=20.09	
Hyperbolic	“@sinh”, “@cosh”, “@tanh”. Example: @tanh(2) = 0.964	
Expression within Range	“@Inrange” Returns a 1 if the number is within the range specified within the function. Example: A1 = 5 • @Inrange(A1,4,7) = 1 • @Inrange(A1,6,10) = 0	

Log Function	Method of Application	View
Expression within Limit	"@Inlimit" Returns a 1 if the number is within the range, on either side of the number, specified within the function. Example: A1 = 5 • @Inlimit(A1,7,2) = 1 • @Inlimit(A1,7,1) = 0	
Expression within Percentage	"@Inpercentage" Returns a 1 if the number is within the percentage, on either side of the number, specified within the function. Example: A1 = 5 • @Inpercentage(A1,8,40) = 1 • @Inpercentage(A1,8,35) = 0	

Trigonometric Functions

All of the trigonometric functions are supported, including inverse and hyperbolic functions:

Trig Function	Method of Application	View
Standard	"@sin", "@cos", "@tan". Example: @cos(pi) =-1 (Radian Angles)	
Inverse	"@asin", "@acos", "@atan". In this case, the number to which the function is being applied must be between -1 and 1. Example: @asin(1) = 1.571 (Radian Angles)	

Trigonometric functions can be calculated using radian, degree or grad units, by selecting the appropriate type from the Angles in drop-down list in the Current Cell group.

Parentheses are required for all logarithmic and trigonometric functions. The capitalization is irrelevant; HYSYS calculates the function regardless of how it is capitalized.

Logical Operators

The Spreadsheet supports Boolean logic. For example, suppose cell *A*1 had a value of 5 and cell *A*2 had a value of 10. Then, in cell *A*3, you entered the formula (+*A*1<*A*2).

The Spreadsheet would return a value of 1 in cell *A*3, since the statement is True (*A*1 is less than *A*2). If the value of either cell *A*1 or *A*2 changes such that the statement is False, cell *A*3 becomes zero.

You can use the following operators:

Boolean	Method of Application
Equal To	“==”
Not Equal To	“!=”
Greater Than	“>”
Less Than	“<“
Greater Than or Equal to	“>=”
Less Than or Equal to	“<=”

IF/THEN/ELSE Statements

The general format of IF/THEN/ELSE statements is:

```
“@if (condition) then (if true) else (if false)”
```

The *condition* is a logical expression, such as “B1 == 15".

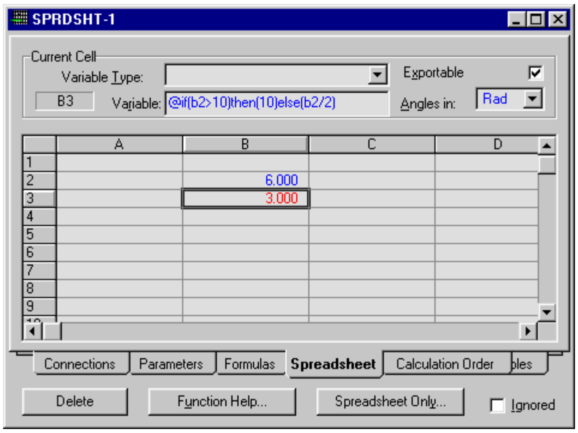
> **You always need to provide an ELSE clause (IF/THEN statements are not accepted).**
>
> **Parentheses are mandatory for IF/THEN/ELSE statements.**

For example, suppose cell B2 contained the number 6. The statement

```
"@if (B2>10) then (10) else (B2/2)"
```

would result in the value 3 being displayed in the cell.

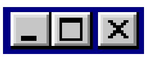
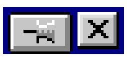
Figure 5.154



5.10.3 Spreadsheet Interface

Importing and Exporting Variables by dragging

You can drag the contents of any cell in the simulation into the Spreadsheet. Simply position the pointer on that field, right-click and drag the value to any cell in the Spreadsheet.

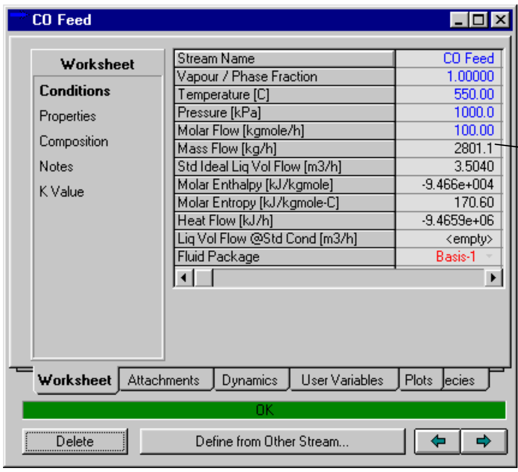
View Type	Description
Non-Modal 	A non-Modal property view has a Minimizing button and Maximizing button in the upper-right had corner, and has a double border. You can drag variables outside a non-Modal property view.
Modal 	A Modal property view has a 'pin' in the upper-right corner, and has a single border. You cannot drag variables outside a Modal property view. Select the pin to convert a Modal property view to a Non-Modal property view.

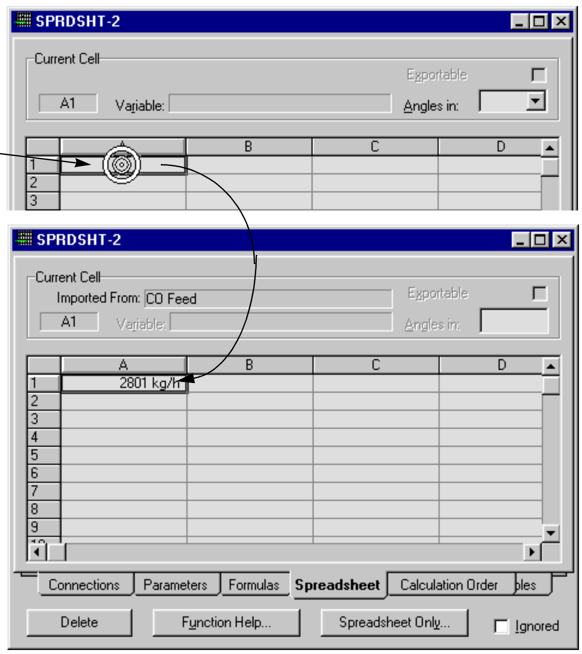
The window from which you are dragging must be unpinned (non-modal). The Spreadsheet window is non-modal by default.

When you drag to a cell in the Spreadsheet, you see the “bulls-eye” cursor. Release the secondary mouse button, and the value is dropped in that cell. In the Imported From field in the Current Cell group (which appears when the cursor is on an imported cell), you see the Object for that particular cell. The Object Variable appears in the Variable field.

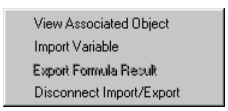
Every time you make a change to (or HYSYS re-calculates) a variable you have placed in the Spreadsheet, your data is updated appropriately.

Figure 5.155





Right-click and drag a variable (in this case, Feed Molar Flow) to a cell in the Spreadsheet. You see the bulls-eye cursor, indicating that you can transfer the variable to that location. Release the mouse button and the variable is transferred. In the **Imported From** cell, you can view the variable source.



Object Inspect menu

You can remove an attachment at any time by positioning the pointer in the appropriate cell, right-clicking and selecting **Disconnect Import/Export** command from the Object Inspect menu.

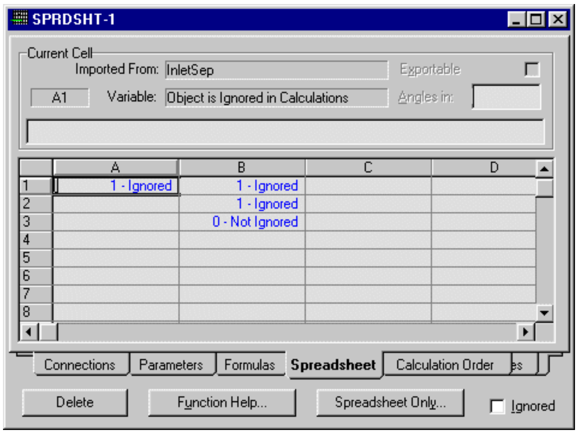
Enumeration in Spreadsheet

Similar to the drag-and-drop importing method, the Ignore checkbox of each operation can be imported onto the Spreadsheet page of the Spreadsheet operation.

For any unit and logical operations, you can right-click on the Ignore checkbox, and drag-and-drop the bulls-eye onto a spreadsheet cell. For an active operation, the cell should then read 0 - Not Ignored. For a disable operation (in other words, the **Ignore** checkbox is selected in the operation property view), the cell should read 1-Ignored.

The ignore status changes automatically when you select or clear the **Ignore** checkbox of the corresponding operation.

Figure 5.156

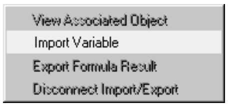


This feature is especially useful when you are working with controllers or actuator fail positions. The ignore status can be used as a number in a formula or in a Boolean expression even though the cell displays the text that reflects the status.

Importing Variables by Browsing

You can also import a variable by positioning the cursor in an empty cell of the Spreadsheet and right-clicking. The Object Inspect menu appears, select the Import Variable command.

Figure 5.157



The Object Inspect menu is accessed by right-clicking the cell in the spread sheet.

Refer to **Section 1.3.9 - Variable Navigator Property View** for more information.

Using the Variable Navigator select the flowsheet variable you want to import to the Spreadsheet. This method of importing variables is similar to the way variables are imported on the Connections tab.

Exporting Formula Results

Variables are exported using the Variable Navigator, or by "dragging" the variable. You can only export Formula Results, in other words, values that appear in red.

There are three ways to export:

- Right-click and drag to the location where you want to export the formula result. You see a bulls-eye cursor indicating that you can export to the current location.

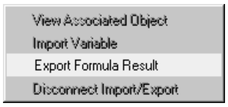
You can only drag the variables to and fro between non-modal property views.

If you export into a field containing a calculated value, you usually get a consistency error, except in the unlikely case that the calculated and exported values are exactly the same.

The export value replaces a specifiable value.

- Right-click and select **Export Formula Result** command from the object inspect menu. Using the Variable Navigator, choose where you want to export the Formula Result.

Figure 5.158



The object inspect menu is accessed by right-clicking the cell in the spread sheet.

- Define an exported variable on the Connections tab by clicking the Add Export button and selecting the export object and variable using the Variable Navigator.

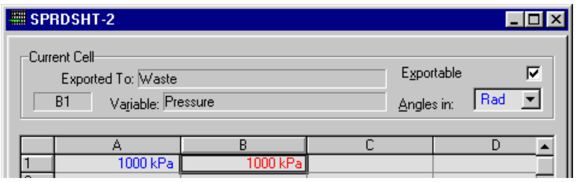
You cannot use the same Spreadsheet cell as both the Target and Source field in calculations.

Similarly, the same Spreadsheet cell cannot act as the Source for more than one field. To work around this, type the cell name with the variable you want to export into a new location in the Spreadsheet, and export the new variable.

Notice that when you export a variable from a Spreadsheet cell, that variable is given the same units as the units of the location to which you exported it.

For example, suppose you wanted to assign the pressure of stream Feed to another stream. In cell *B*1, enter the formula +*A*1, and then export the contents of the cell to the pressure cell of the appropriate stream, using one of the methods outlined above.

Figure 5.159



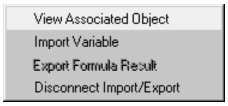
Because the contents of cell A1 cannot be both an import and export, the formula +A1 is entered in cell B1. cell B1 is then exported to the Waste pressure.

For this simple example, you could use the Set operation. For more complex situations, you must use the Spreadsheet.

View Associated Object

You can view an object associated with a specific cell by right-clicking and selecting the View Associated Object.

Figure 5.160



The Object Inspect menu is accessed by right-clicking the cell in the spread sheet.

For instance, if you dragged the temperature of a stream from the Worksheet into the Spreadsheet, the associated object would be that stream. When you select View Associated Object, you are taken to the property view for that stream.

You can also view the associated object of an imported cell, by double-clicking on that cell.

If there is no object associated with the current cell, this menu selection is disabled.

5.10.4 Spreadsheet Tabs

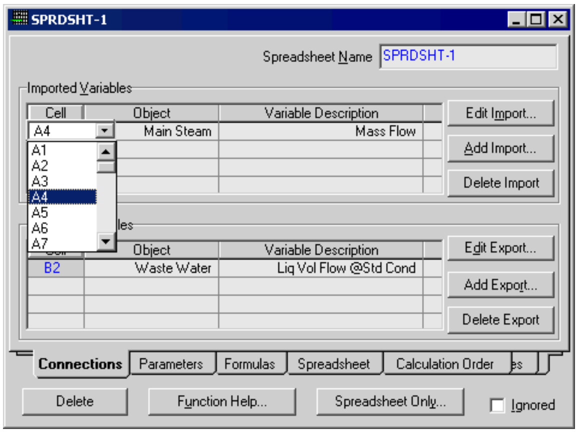
Connections Tab

On the Connections tab, you can add, edit, and delete Imports and Exports. As mentioned earlier, you can also import and export variables by dragging to and from the Spreadsheet.

Refer to **Section 1.3.9 - Variable Navigator Property View** for more information.

To add an import, click the **Add Import** button, and choose the variable using the Variable Navigator. In the **Cell** column, type or select from the drop-down list the Spreadsheet cell to be connected to that variable. When you move to the **Spreadsheet** tab, that variable appears in the cell you specified. An example is illustrated in the following figure.

Figure 5.161



You can edit or delete an import by positioning the cursor in the appropriate row, and clicking the **Edit Import** or **Delete Import** buttons. Adding, editing, and deleting Exports is performed in a similar manner. You can also edit the Spreadsheet Name on this tab.

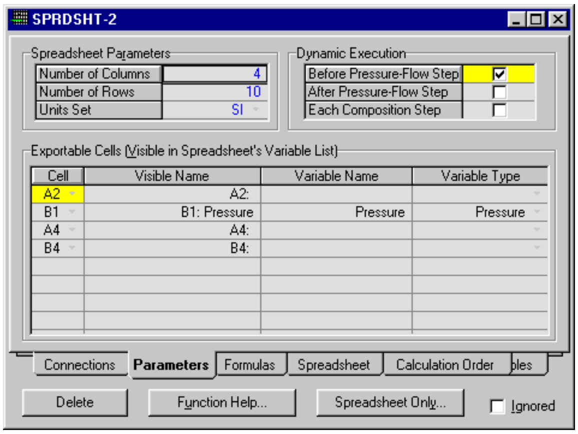
Parameters Tab

On the Parameters tab of the Spreadsheet property view, you can set the dimensions of the Spreadsheet and choose a Unit Set.

Parameters	Description
Number of Columns and Rows	You can set the dimensions of the Spreadsheet. Notice that if you set the dimensions of the Spreadsheet smaller than what is already specified, you permanently delete the contents of cells which are removed. For instance, the contents of cell A4 and B4 are deleted when you set the Number of Rows to 3.
Units	You can choose a Unit Set for the Spreadsheet. All values in the Spreadsheet appear using units from the set you have chosen.

Refer to **Section 12.3.1 - Units Page** in the **HYSYS User Guide** for more information about Unit Set.

Figure 5.162



Exportable Cells

Prior to explaining how the Exportable cells are created, the difference between exporting from the Spreadsheet (assigning a value from the Spreadsheet to a Process Variable) or importing from the Spreadsheet (accessing a Spreadsheet variable from another object) must be explained.

Results that are exported from the Spreadsheet to a specifiable process variable can only be connected once. In other words, the same cell cannot be connected to two process variables.

However, locations in the program which can import from the Spreadsheet (for example, PID Controller Cascade Source, Adjust Target Variable or Databook Variable) can access any cell, including those which are being exported to a flowsheet process variable. The Exportable Cells list has been created to allow objects which use the Variable Navigator to access variables associated with the Spreadsheet.

The Exportable Cells group displays all cells which can be exported (including those which have been exported). The Visible Name, Variable Name, and Variable Type either displays the information you have specified for the associated cell on the Spreadsheet itself, or contains the information appropriate to the process variable that the cell has been exported to. In the former case, this information is modifiable; you can change it here or on the Spreadsheet itself. In the latter, you cannot modify the information as it is set by the process variable the cell has been exported to.

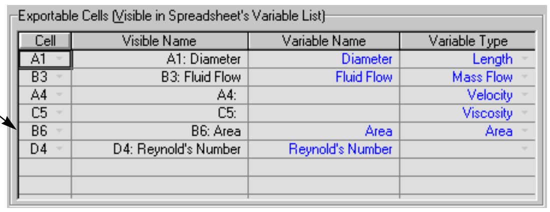
> **The Visible Name and Variable Name columns display variables which can be exported. The fact that a variable appears in this list does not necessarily mean that the variable has been exported.**
>
> **When you access the Spreadsheet as the Object (for example, through the Variable Navigator), the contents of the Visible Name cell appear in the Variable List.**

For instance, if you export a Spreadsheet value to the Separator Valve Opening cell, the Variable Name and Variable Type are Valve Opening and Percent, respectively.

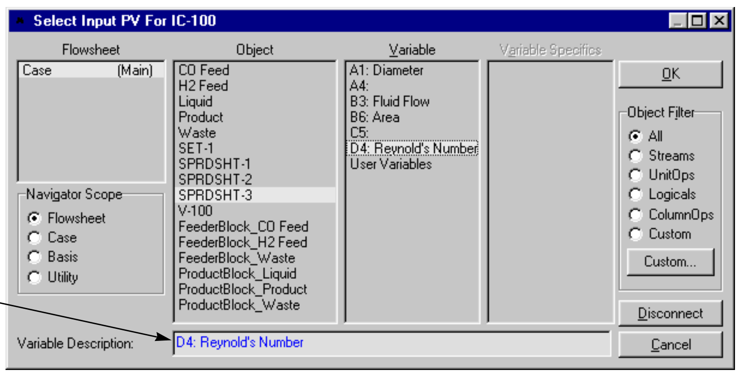
You can edit the Variable Name and Variable Type for all non-exported variables that appear in the list

Figure 5.163

There are three variables in the Variable List for SPRDSHT-1 corresponding to cells A1, B3, and B6. Notice that the Variable Names were added manually.



When you use the Variable Navigator and select SPRDSHT-1 as the Object you see cells A1, B3, B6, A4, C5, and D4 in the Variable List.

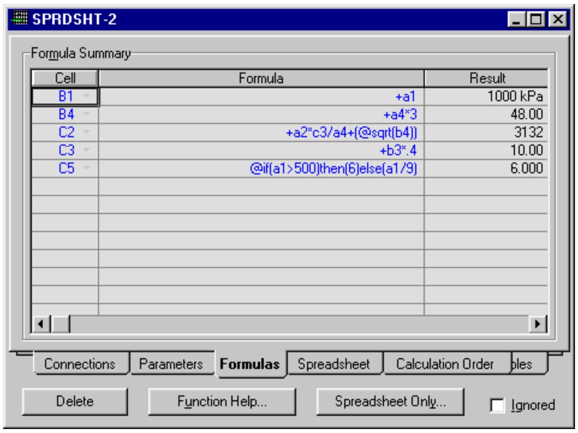


Spreadsheet variables attached to the Controller, Adjusts, and Databook are not exported, but are imported from that Object.

Formulas Tab

The Formulas tab displays a summary of all the formulas included in your spreadsheet. The table lists the name of the cell the formula is located in, the formula and the result of the formula.

Figure 5.164



Spreadsheet Tab

The Spreadsheet tab, with the labelled rows and columns, is similar to conventional Spreadsheets.

From this tab, you can import and export variables, disconnect imports/exports, view associated object property views, define formula expressions, and modify variable names.

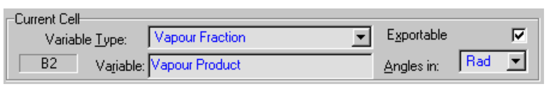
Spreadsheet Function	For More Information
Importing and Exporting	See sections: • **Importing and Exporting Variables by dragging** • **Importing Variables by Browsing**
Associated Object Views	Refer to section **View Associated Object**.
Formula Expressions	Refer to **Section 5.10.2 - Spreadsheet Functions**.
Variable Names	Refer to section **Exportable Cells**.

Current Cell Group

The Current Cell group displays information specific to the contents of the highlighted cell. For all cases, the Current Cell location appears.

Cell containing a Formula or non-imported specifiable value

Figure 5.165



The Variable Type and Variable Name are shown. You can choose a new Variable Type from the drop-down list, and you can edit the Variable name.

> **The Variable Type sets the units for the Spreadsheet cell. For example, the SI units for Variable Type Area are m^2.**

Cells containing a formula or a non-imported specifiable value are automatically added to the Variable list on the Parameters tab; the **Exportable** checkbox is selected.

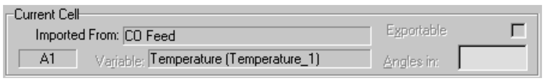
Cell containing an Export

Figure 5.166

The object and variable to which the contents of the cell were exported are shown. The **Exportable** checkbox is selected in this case. You cannot change the Variable Name, since it is a HYSYS default.

Cell containing an Import

Figure 5.167

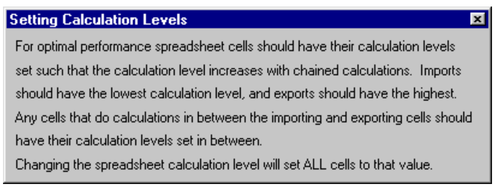


The object and variable from which the contents of the current cell were imported are shown. You cannot change the Variable name, since it is a HYSYS default.

Calculation Order Tab

The Calculation Order tab allows you to set the calculation level of each of the cells in the spreadsheet. Click the Calculation Order Help button to view the rules for setting the calculation levels.

Figure 5.168



User Variables Tab

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables tab enables you to create and implement your own user variables for the current operation.

Function Help and Spreadsheet Only buttons

Refer to **Section 5.10.2 - Spreadsheet Functions** for more information.

Clicking the Function Help button allows you to view the available Spreadsheet Functions and Expressions. Notice that this Help Window has three tabs:

- Mathematical Expressions
- Logical Expressions
- Mathematical Functions

Click the Spreadsheet Only button to view just the Spreadsheet cells in a separate window. This feature is useful when you have completely set up the Spreadsheet, and you only want to view the cell results.

5.11 Stream Cutter

The stream cutter is an object that allows you to switch the fluid package of a stream anywhere in the flowsheet. This concept of changing fluid package is called fluid package transition.

HYSYS automatically adds a stream cutter operation in between two objects on the PFD property view, where a switch in fluid package occurs.

5.11.1 Stream Cutter Property View

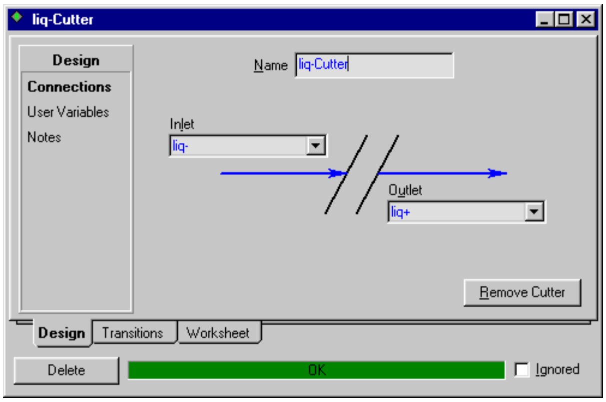
To add a Stream Cutter to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Logicals** radio button.
3. From the list of available unit operations, select **Stream Cutter**.
4. Click the **Add** button.

The Stream Cutter property view appears.

Figure 5.169



The Stream Cutter property view contains the following tabs:

- Design
- Transitions
- Worksheet

Changing Fluid Package

Currently, there are several methods for you to change the fluid package of objects:

- using a drop-down list from the unit operation property view.
- right-clicking on a selected group of operations in the PFD property view.
- changing the flowsheet fluid package in the basis environment.

If you add an operation and change its fluid package before any streams are connected to the operation, then connect empty streams (default fluid package, not connected to anything and empty composition) to the operation, HYSYS changes the empty stream's fluid package to the operation's fluid package.

If a stream connected to an operation with a specified fluid package has one of the following:

- a specified fluid package.
- a connection to another operation.
- its composition specified.

HYSYS adds a stream cutter between the stream and the operation the stream is attached to.

HYSYS does not allow fluid package transitions in electrolytic flow sheets or inside of column flow sheets. Fluid package transitions are allowed only in standard flow sheets.

It is recommended to have all the fluid package specifications in place before switching to dynamics.

Changing the Fluid Package in the Unit Operation Property View

You can change the fluid package of an operation using the Fluid Package drop-down list on the Connections page of the Design tab for the Operation property view.

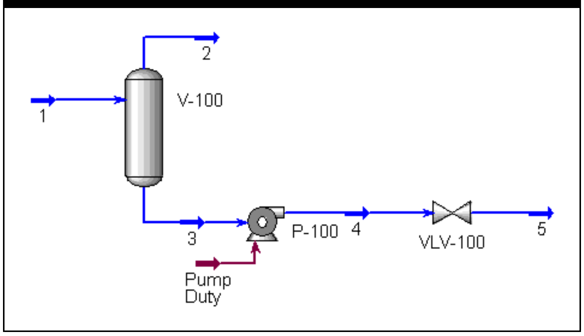
You can change the fluid package of a stream on the Stream property view - Worksheet tab - Conditions page.

In this method, HYSYS propagates the new fluid package specifications to connected operations and streams. This propagation stops when it encounters one of the following:

- a fluid package (either on an operation or stream) that you have already specified.
- an operation with more than one feed or product.
- a template or column.
- an existing stream cutter.

For example, consider a separator with its liquid stream connected to a pump, which is in turn connected to a valve. All objects are using FP1, the default for the flowsheet.

Figure 5.170



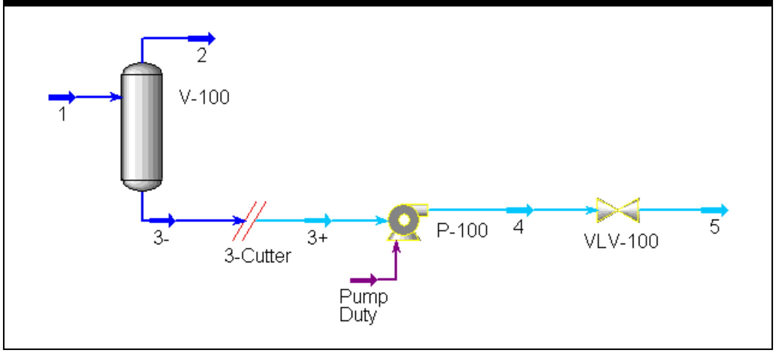
If you specify the fluid package of the valve to FP2, HYSYS first propagates downstream. HYSYS finds a single stream, 5, with a fluid package at default status, which is FP1. So HYSYS calculates and changes stream 5 to the FP2 fluid package setting.

The propagation calculation basically steps along. The valve calculates into stream 4, stream 4 calculates into the pump, the pump calculates into stream 3, and so forth.

Next HYSYS propagates upstreams. The stream 4 also has a default fluid package setting. So HYSYS calculates, changing stream 4 to FP2 setting. The propagation goes along stream 4 until it encounters the pump. The pump has a default fluid package setting, and only a single inlet and a single outlet (energy streams are not considered because they have no dependence on fluid package). HYSYS calculates and changes the pump fluid package setting to FP2. The propagation continues to the pump's inlet stream 3. The stream has a default fluid package setting of FP1, so HYSYS calculates and moves upstream to the separator. The separator has multiple outlet streams. So when HYSYS encounters the separator the propagation function stops, and HYSYS adds a cutter.

Since the propagate calculation has already calculated stream 3 with fluid package FP2, HYSYS generates another stream called 3- with FP1 as the fluid package, and renames stream 3 to 3+, which has FP2 as the fluid package.

Figure 5.171



At the point where the propagation stopped, a stream cutter is automatically added to the flowsheet between the operations in question, and a fluid package transition is added to the cutter's transition collection. This fluid package transition automatically chooses its component mappers to be the default mapper of the appropriate map collection. The default mapper can be changed by entering the Basis environment, opening the Simulation Basis Manager property view, and selecting the Component Maps tab.

The fluid package transition function, however, does not automatically choose a transfer basis. You must make this decision. The status window should have a missing required info error for the stream cutter stating **Transitions not ready**.

Figure 5.172

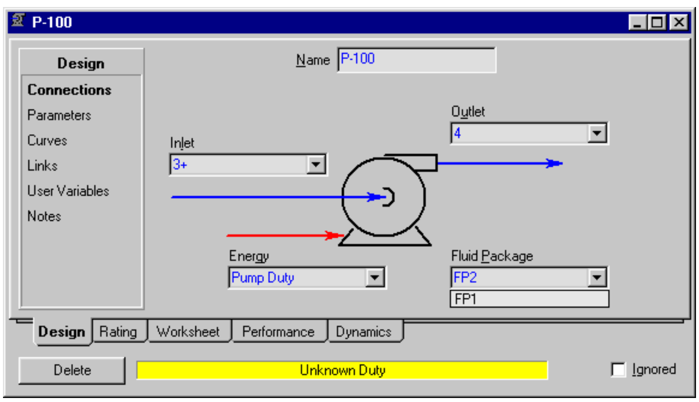
Refer to the **Fluid Pkg Page** section for more information.

By double-clicking the statement in the status window, the stream cutter property view opens already on the Fluid Pkg page of the Transitions tab.

In addition to automatically adding stream cutters where transitions between fluid packages are occurring, HYSYS can determine when stream cutters are no longer needed and prompts you to decide if they should be removed.

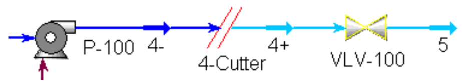
For instance, continuing on from the previous example, change the pump's fluid package back to FP1.

Figure 5.173



First consider the downstream propagation. This propagation goes through the pump outlet stream 4 and finds the valve. Since this is where you made your first fluid package specification, the status of the fluid package is specified and propagation stops, and a stream cutter is automatically added (again the fluid package transition requires a transfer basis).

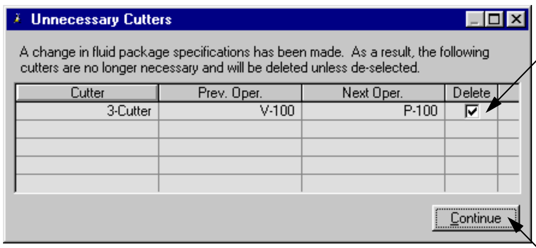
Figure 5.174



Now consider the upstream propagation. The FP1 setting propagates through the inlet of the pump and encounters the stream cutter along stream 3+. The propagation stops according to the rules. This stream cutter contains the fluid package transition between FP1 and FP2, but now both sides are FP1. Since you have not added any other transitions to the stream cutter, the stream cutter only contains a fluid package transition. This fact combined with the fact that each side of the cutter has the same fluid package, HYSYS can assume the stream cutter is no longer needed.

A property view appears at this point listing all unnecessary stream cutters, and you can select which ones, if any, to delete.

Figure 5.175



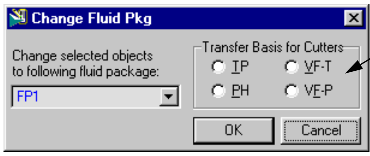
Select or clear the checkbox in the **Delete** column to remove or keep the stream cutters.

Click the **Continue** button to remove the selected stream cutters.

Changing the Fluid Package Using the Object Inspect Menu

In order to bypass the propagation rules used by the unit operation property view method, you can change the fluid package of only certain operations of your choice. Select the operations you want from the PFD (like you would for copy/paste), then right-click to open the object inspect menu. Select the Change Fluid Package command from the menu, and the Change Fluid Pkg property view appears.

Figure 5.176

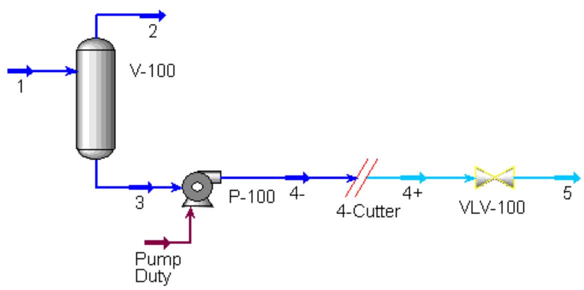


You can select the transfer basis using the radio buttons in the Transfer Basis for Cutters group.

Using the drop-down list, you can select the fluid package you want for the selected objects.

Consider the separator-pump-valve train defined in the previous section. By the end, you were left with the separator and pump using FP1 and the valve using FP2, with a cutter between the pump and valve.

Figure 5.177



At this state, the flowsheet contains two individual fluid package specifications, one on the valve and the other on the pump. Depending on the flowsheet configuration and the type of operations involved, it is important to note that by using the unit operation property view method you could end up with a less-than-desirable flowsheet with stream cutters where you didn't want them. These hassles can be avoided by using object inspect, and the same result achieved in the previous section with two specifications can be achieved with one.

To achieve the results in the above figure using only one specification, you use the object inspect menu. First select the valve and attached streams on the PFD. Then right-click to bring up the object inspect menu, and select the Change Fluid Pkg command. The Change Fluid Pkg property view appears. Select FP2 from the drop-down list, and a transfer basis from the radio button. Propagation is suppressed, so only streams 4 and 5 and the valve is switched to FP2 fluid package. A stream cutter is also added with default maps and the selected transfer basis between valve inlet stream 4 and the pump outlet stream 4.

If you use the object inspect method to change fluid packages, and you happen to have a heat exchanger or LNG selected, all exchange sides of the exchanger in question get switched. There is no way of picking and choosing particular exchanger sides, except from the actual exchanger and LNG property views.

Changing the Fluid Package from the Basis Environment

Refer to **Section 2.2 - Fluid Packages Tab** in the **HYSYS Simulation Basis** guide for more information.

You can change the fluid package of the default fluid package setting in the Basis environment. Open to the Simulation Basis Management property view, and click on the Fluid Pkgs tab. Select a different fluid package using the Default Fluid Pkg drop-down list. Notice that when you change the default fluid package setting, only objects whose fluid packages are still at default setting get switched to the new fluid package default.

5.11.2 Design Tab

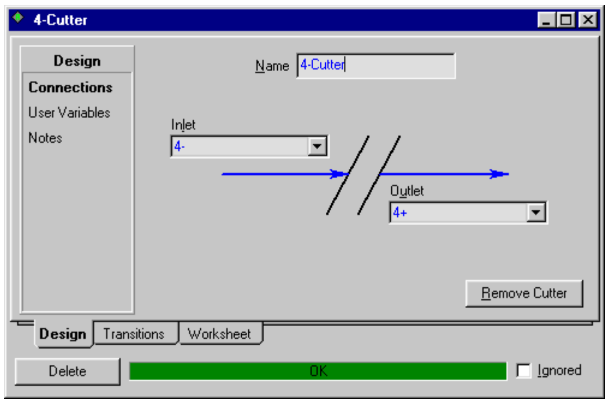
The Design tab contains the following pages:

- Connections
- User Variables
- Notes

Connections Page

You can select the inlet and outlet streams of the stream cutter on this page. You can change the name of the stream cutter in the Name field.

Figure 5.178



You can click on the Remove Cutter button to uncut the attached streams. The Remove Cutter button is available only after a stream has been specified in both the inlet field and outlet field.

The difference between the Remove Cutter button and Delete button is that the Remove Cutter button maintains upstream and downstream connections.

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor, where you can record any comments or information regarding the operation or to your simulation case in general.

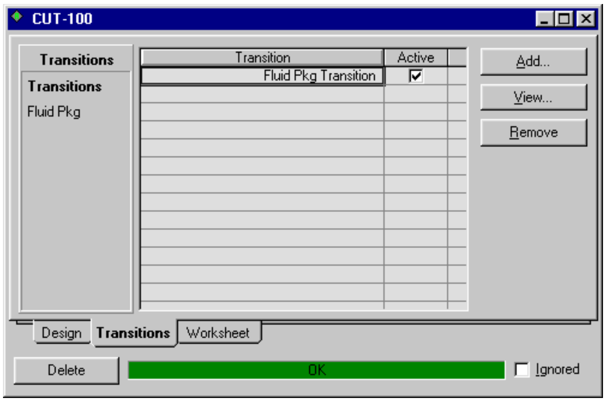
5.11.3 Transitions Tab

The Transitions tab contains the following pages:

- Transitions
- Fluid Pkg

The Fluid Pkg page is available only after you add the option into the Transitions page, as shown in the figure below.

Figure 5.179

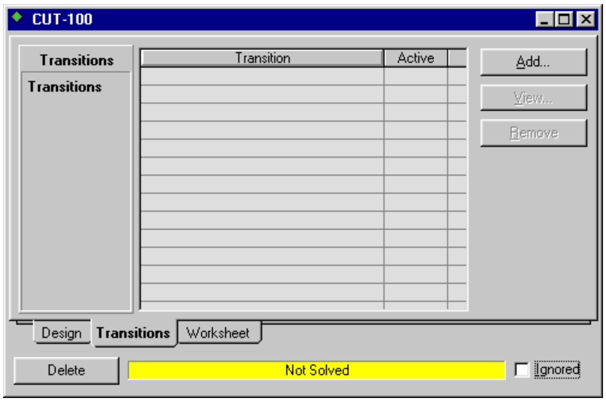


The Fluid Pkg page is automatically available, if the stream cutter was generated by HYSYS to perform fluid package transition.

Transitions Page

The Transitions page contains a table and three buttons: Add, View, and Remove. When no transition type has been selected, only the Add button is available for use, and the table is blank as shown in the figure below.

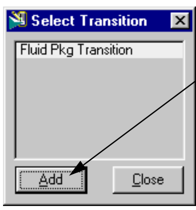
Figure 5.180



Adding a Transition

Click the Add button to open the Select Transition property view.

Figure 5.181



Select the transition type you want from the list, and click the Add button.

Click the **Close** button to close the property view when you are done.

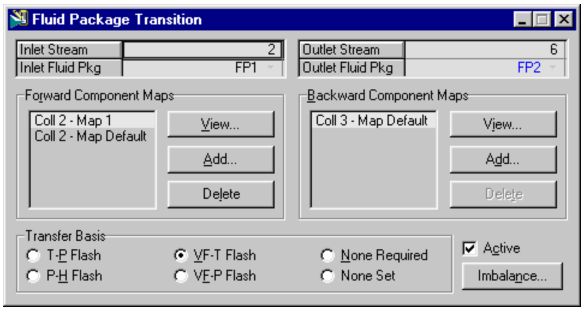
From this property view you can add the transition type you want the stream cutter to perform. Currently there is only one transition type available: fluid package.

Once you have selected a transition type and return to the transition page, the **View** and **Remove** buttons are available for use.

Viewing a Transition

Select a transition type from the list, and click the **View** button to open a property view that contains more detailed information about the transition type and its functions. The figure below shows a fluid package transition property view.

Figure 5.182



The Fluid Package Transition property view consists of the following objects:

Depending on how the stream cutter is added and how the transition type is defined, the field texts can be black to indicate non-changeable values, or blue to indicate changeable values.

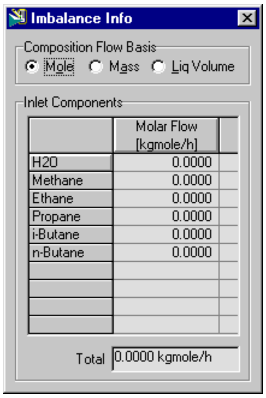
Object	Description
Inlet Stream Field	Displays the name of the stream going into the stream cutter.
Inlet Fluid Pkg Field	Displays the fluid package being used by the inlet stream.
Outlet Stream Field	Displays the name of the stream coming out of the stream cutter.
Outlet Fluid Pkg Field	Displays the fluid package being used by the outlet stream.

Refer to **Section 6.3 - Component Map Property View** in the **HYSYS Simulation Basis** guide for more information about the Component Map property view.

Refer to **Section 6.3 - Component Map Property View** in the **HYSYS Simulation Basis** guide for more information about the Component Map property view.

Object	Description
Forward Component Map Group	Lists the component maps that are used if composition is passed from the inlet stream to the outlet stream. • Click the **Add** button to add another component map. • Click the **View** button to open the Component Map property view, that contains more information regarding the selected component map. • Click the **Delete** button to delete the selected component map. You cannot delete the default component map. The list does not affect the simulation, only the selected map does. The map determines how components are mapped across. The maps are necessary only when dealing with component lists that are different. The maps tell HYSYS how to transfer the compositions.
Backward Component Map Group	Lists the component maps that are used if composition is passed from the outlet stream to the inlet stream. • Click the **Add** button to add another component map. • Click the **View** button to open the Component Map property view, that contains more information regarding the selected component map. • Click the **Delete** button to delete the selected component map. You cannot delete the default component map. A component map is necessary when, you have a fluid package FP7 with seven components and another fluid package FP6 with six components. HYSYS cannot just pass the mole fractions of the seven components to the other fluid package, because there are only six slots to put the information in. The component map tells HYSYS how to transfer the mole fraction values.
Transfer Basis Group	Contains six transfer basis types available for the fluid package transition tool. You must select one of the transfer basis radio button.The transfer basis types are: • **T-P Flash**. Transfers temperature or pressure. • **P-H Flash**. Transfers pressure and enthalpy • **VF-T Flash**. Transfers vapour fraction and temperature. • **VF-P Flash**. Transfers vapour fraction and pressure. • **None Required**. Select this radio button when there is no need for a transfer basis. Use this transfer only for energy streams. • **None Set**. This is the default setting. Stream cutter does not solve if the radio button is selected.

Object	Description
Active Checkbox	This checkbox is the same checkbox from the table on the Transitions page. You can select or clear this checkbox to activate or deactivate the transition. When the Active checkbox changes in this property view, the changes also occur in the table on the Transitions page.
Imbalance Button	Click this button to open the Imbalance Info property view. The Imbalance Info property view displays any mole, mass, or liquid volume imbalance that can occur when switching fluid package. The property view is pertinent for fluid package transition involving two different components list.

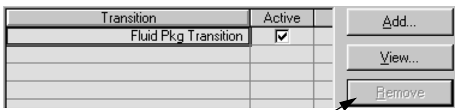


Imbalance Info property view

Removing a Transition

Select the transition type you want to remove from the table, and click the Remove button. If the transition type is currently being used in the stream cutter, the Remove button automatically becomes unavailable when you select the transition from the table.

Figure 5.183



The selected transition is Fluid Pkg, which is also the transition being used in the stream cutter. So the Remove button is disabled.

When you remove a transition from the table, the page associated with the transition is also removed from the Transition tab.

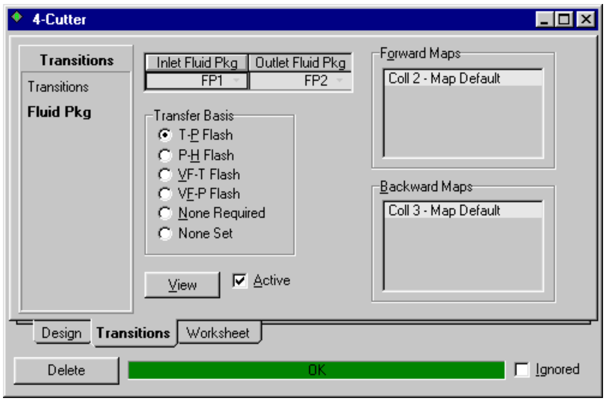
Activate a Transition

Select the checkbox in the **Active** column to activate the associated transition type. Clearing the checkbox associated with the transition type, deactivates the transition but does not remove the transition from the table.

Fluid Pkg Page

The Fluid Pkg page contains information about the fluid package transition. You can also change the transfer basis on this page.

Figure 5.184



Depending on how the stream cutter is added and how the transition type is defined, the field texts can be black to indicate non-changeable values, or blue to indicate changeable values.

The following table lists and describes the objects on this page.

Object	Description
Inlet Fluid Pkg Field	Displays the fluid package being used by the inlet stream.
Outlet Fluid Pkg Field	Displays the fluid package being used by the outlet stream.

Refer to **Section 6.3 - Component Map Property View** in the **HYSYS Simulation Basis** guide for more information about the Component Map property view.

Refer to **Section 6.3 - Component Map Property View** in the **HYSYS Simulation Basis** guide for more information about the Component Map property view.

Object	Description
Forward Maps Group	Lists the component maps that are used if composition is passed from the inlet stream to the outlet stream. • Click the **Add** button to add another component map. • Click the **View** button to open the Component Map property view, that contains more information regarding the selected component map. • Click the **Delete** button to delete the selected component map. You cannot delete the default component map. The list does not affect the simulation, only the selected map does. The map determines how components are mapped across. The maps are necessary only when dealing with component lists that are different. The maps tell HYSYS how to transfer the compositions.
Backward Maps Group	Lists the component maps that are used if composition is passed from the outlet stream to the inlet stream. • Click the **Add** button to add another component map. • Click the **View** button to open the Component Map property view, that contains more information regarding the selected component map. • Click the **Delete** button to delete the selected component map. You cannot delete the default component map. A component map is necessary when, you have a fluid package FP7 with seven components and another fluid package FP6 with six components. HYSYS cannot just pass the mole fractions of the seven components to the other fluid package, because there are only six slots to put the information in. The component map tells HYSYS how to transfer the mole fraction values.
Transfer Basis Group	Contains six transfer basis types available for the fluid package transition tool. You must select one of the transfer basis radio button.The transfer basis types are: • **T-P Flash**. Transfers temperature or pressure. • **P-H Flash**. Transfers pressure and enthalpy • **VF-T Flash**. Transfers vapour fraction and temperature. • **VF-P Flash**. Transfers vapour fraction and pressure. • **None Required**. Select this radio button when there is no need for a transfer basis. Use this transfer only for energy streams. • **None Set**. This is the default setting. Stream cutter does not solve if the radio button is selected.

Object	Description
Active Checkbox	This checkbox is the same checkbox from the table on the Transitions page. You can select or clear this checkbox to activate or deactivate the transition. When the Active checkbox changes in this property view, the changes also occur in the table on the Transitions page.
View Button	Click this button to open the transition type property view.

Refer to the section on **Viewing a Transition** for more information.

5.11.4 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation. The PF Specs page is relevant to Dynamics cases only.

5.12 Transfer Function

The Transfer Function block is a logical operation which takes a specified input, and applies the chosen transfer function to produce an output. A typical use of the Transfer Function is to apply disturbances to a process, such as varying the temperature of a feed stream without having to add a disturbance manually. It is also useful to simulate a unit for which you know the response characteristics (gain, damping factor, period) but not the actual equations involved.

The following Transfer Functions are available:

- First & second order lead
- First & second order lag
- Second order lag / sine wave
- Delay
- Integrator
- Ramp
- Rate Limiter

The second order lag can be defined either as a series of two first-order lags, or as a single explicit second order lag.

Combinations of the above functions may be used to produce the desired output. The combined transfer function is as follows:

$$G(s) = Lead1(s)Lead2(s)Lag1(s)Lag2(s)W(s)D(s)R(s) \tag{5.33}$$

The input X(s) is multiplied by the transfer function to obtain the output. Notice that the input (or Process Variable Source) is optional; you can use a fixed value as the input.

$$Y(s) = G(s)X(s) \tag{5.34}$$

The transfer function is defined here in the Laplace Domain (using the Laplace Variables). When in the Laplace Domain, the overall transfer function is simply the product of the individual transfer functions.

The Laplace Transfer Function must be converted to a real-time function in order to be meaningful for a dynamic simulation. For instance, the Laplace Transform for the sine function is:

$$G = \frac{\omega}{s^2 + \omega^2} \tag{5.35}$$

When converted to the time domain by taking the inverse Laplace, we obtain:

$$f(t) = \sin\omega t \tag{5.36}$$

5.12.1 Transfer Function Property View

There are two ways that you can add a Transfer Function to your simulation:

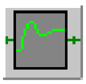
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Logicals** radio button.
3. From the list of available unit operations, select Transfer Function Block.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

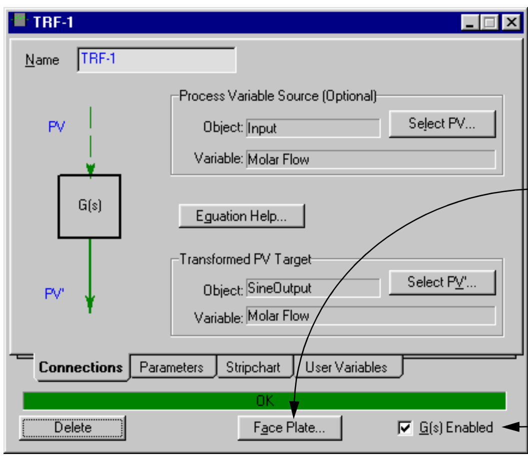
   You can also open the Object Palette by pressing **F4**.
2. Double-click the **Transfer Function** icon.



Transfer Function icon

The Transfer Function property view appears.

Figure 5.185



Click the **Face Plate** button to access the Transfer Function face plate property view.

Select the **G(s)Enabled** checkbox to combine more than one specified functions in the **Parameters** tab.

5.12.2 Connections Tab

The following information is shown on the Connections tab:

Refer to **Section 1.3.9 - Variable Navigator Property View** for more information.

Input Required	Description
Name	The name of the Transfer Function Block.
Process Variable Source	A stream or operation. You can select the PV Object and Variable by clicking the Select PV button. The Process Variable is optional. If you do not specify a PV, enter a constant PV on the Parameters tab.
Transformed PV Target Object	A stream or operation. Select the PV Object and Variable by clicking the Select PV' button. The PV Target is not required to have the same variable type as the PV Source.

You can click the **Equation Help** button to view the Transfer Function equations.

5.12.3 Parameters Tab

The Parameters tab allows you to define the entire transfer function G(s), by defining the integrator, delay, lag, lead, and 2nd order transfer functions.

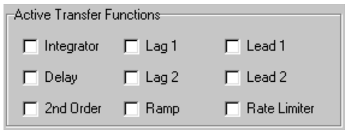
The Parameters tab contains the following pages:

- Configuration
- Integrator
- Delay
- Lag
- Lead
- 2nd Order
- Ramp
- Rate Limiter

The last seven pages allow you to define the different transfer function terms. Each of these pages contains the Active Transfer Function group, which consists of a number of checkboxes corresponding to the available components of the Transfer Function. By selecting the appropriate checkbox, you can include that term in the overall Transfer Function. When you activate individual functions on the Integrator/Delay/Lag/Lead/

2nd Order/Ramp/Rate Limiter pages, the appropriate checkboxes are then selected in this group.

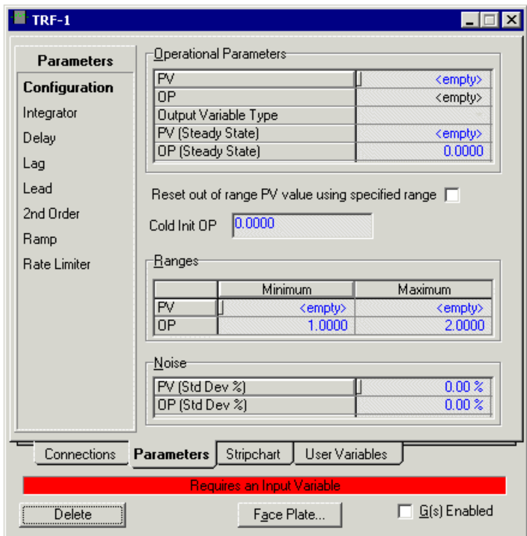
Figure 5.186



Configuration Page

The Configuration page allows you to define the Process and Output Variable limits.

Figure 5.187



The Operational Parameters group contains the following parameters:

Parameter	Description
PV	The value of the PV input (Process Variable or constant PV) is shown in this field. • If you did not define a PV source in the Connections tab, then you must specify a constant PV value in this field. Notice the text is blue in colour, indicating you can change this value. • If you defined a PV source on the Connections tab, then this field displays the PV Input. Notice the text is black in colour, indicating it is a HYSYS calculated value.
OP	Displays the calculated value of the PV Output.
Output Variable Type	Displays the OP variable type.

Selecting the **Reset out of range PV value using the specified range** checkbox tells HYSYS to reset the PV input value whenever the PV value deviates outside the specified range. The specified range can be entered in the Ranges group. The reset value is the value entered in the **PV** field from the Operational Parameters group.

The Reset out of range PV value using the specified range checkbox is not available if a PV source is defined on the Connections tab.

The Ranges group contains the following parameters:

Parameter	Description
PV Minimum and Maximum	Enter the percent range of the Transfer Function Input. These percent values define the range of the input value; regardless of the varying input value from the source or Transfer Function parameters, the input value always stays in this range. This percent range affects the Noise and sine wave amplitude.
OP Minimum and Maximum	Enter the range of the Transfer Function Output. These values define the range of the output; regardless of the input or Transfer Function parameters, the output always stays in this range. This range affects the Noise and sine wave amplitude.

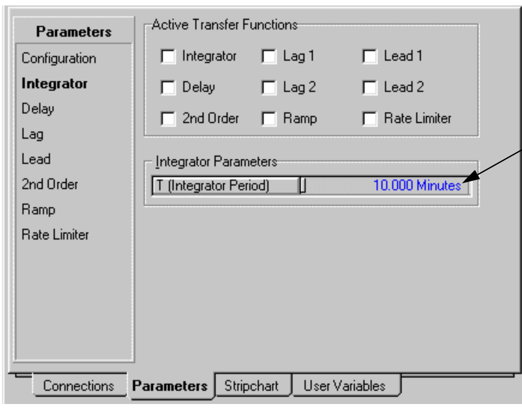
The Noise group contains the following parameters:

Parameter	Description
PV (Std Dev %)	Enter the Standard Deviation of the input noise as a percentage of the PV Range.
OP (Std Dev %)	Enter the Standard Deviation of the output noise as a percentage of the PV Range. The noise follows a normal distribution.

Integrator Page

The Integrator page consists of the Active Transfer Function group and Integrator Parameters group.

Figure 5.188



You can change the integrator period value in this field.

The Integrator Transfer Function requires only one parameter, the T (Integrator Period) located in the Integrator Parameters group:

$$G = \frac{1}{Ts} \qquad (5.37)$$

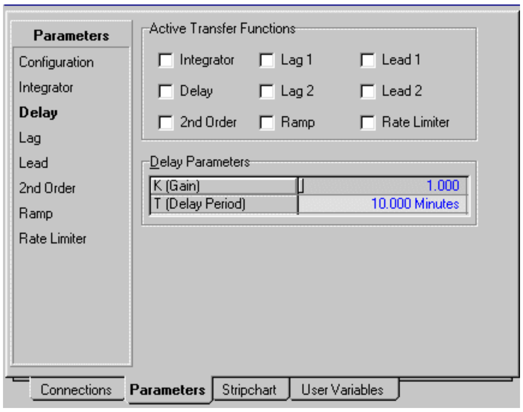
The unit step response of the Integrator Transfer Function is

$$f(t) = \frac{t}{T} \qquad (5.38)$$

Delay Page

The Delay Page consists of the Active Transfer Function group and Delay Parameters group.

Figure 5.189



The Delay Parameters group contains of two input fields the K (Gain) and T (Delay Period) that are two parameters of the Delay Equation.

The Delay Equation is defined as:

$$G = Ke^{-t_o s} \tag{5.39}$$

where:

t_o = *dead time*

The inverse Laplace Transform of the Delay Equation multiplied by the general function *F(s)* is equal to $Kf(t\text{-}t_o)$. This is shown below:

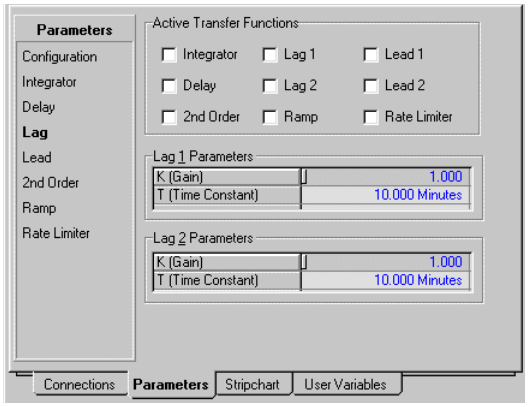
$$L^{-1}(Ke^{-t_o s}F(s)) = Kf(t - t_o) \tag{5.40}$$

Delay can be used in combination with the other Transfer function terms, by selecting the **Delay** checkbox in the Active Transfer Function group.

Lag Page

The Lag page allows you to simulate the response of a first-order or second-order lag. A second order lag can be defined on the Lag page creating two first-order lags.

Figure 5.190



The Lag page contains the Lag 1 Parameters group and Lag 2 Parameters group; with each group defining a single-order lag transfer function. The group contains two fields the K (Gain) and T (Time Constant).

The Lag Equation is defined as follows:

$$G = \frac{K}{Ts + 1} \tag{5.41}$$

where:

G = transfer function

K = gain

T = time constant (t)

s = Laplace Transform variable

The time constant is the time required for the response to reach 63.2% of its final value.

The unit step response of the Lag Equation is:

$$K\left(1 - e^{-\frac{t}{T}}\right) \tag{5.42}$$

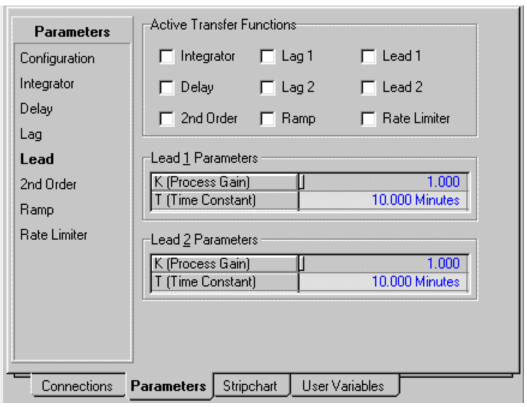
Refer to the section on the **2nd Order Page** for more information.

A second-order Lag may also be defined on the 2nd Order page. This second-order Lag is defined using variables K, T, and ξ .

Lead Page

The Lead page allows you to define either a first or second order Lead transfer function. This is done via the two groups the Lead 1 Parameters group and Lead 2 Parameters group. Both groups allow for the definition of the two terms of the Lead Equation K and T.

Figure 5.191



The Lead Equation is defined as:

$$G = K(Ts+1) \tag{5.43}$$

where:

K = gain

T = time constant

The Inverse Laplace Transform of the Lead Equation multiplied by the general function F(s) is:

$$L^{-1}[K(Ts+1)F(s)] = K\left[\frac{df(t)}{dt}T + f(t)\right] \tag{5.44}$$

A first or second-order Lead can be simulated by making one or both Lead Parameters active. You can make a set of K and T active by selecting the **Lead** checkbox in the Active Transfer Functions group.

The response is an exponential curve of the following form:

$$Flow(t) = K\left(1 - e^{-\frac{t}{T}}\right) \tag{5.45}$$

where:

K = process gain

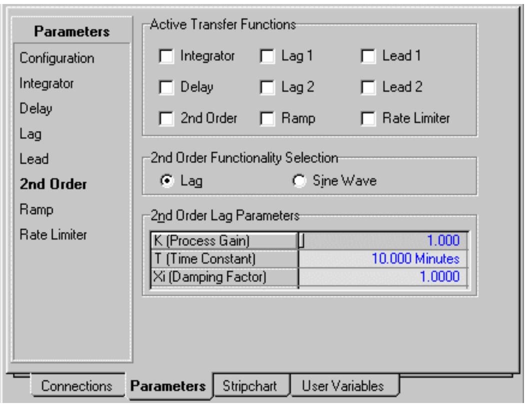
T = time constant

The time constant is the time required for the response to reach 63.2% of its final value. In this case, the time constant is 600s (10 minutes), so the response should have a value of about 63 kgmole/h 10 minutes after the step change in Input is introduced. This is illustrated in the Strip Chart.

2nd Order Page

The 2nd Order page allows you to define either the second order or sine wave function.

Figure 5.192



Standard Second Order

Select the Lag radio button in the 2nd Order Functionality Selection group, which is used to simulate the response of a standard Second Order process.

The Second Order Lag is defined as:

$$G = \frac{K}{T^2 s^2 + 2T\xi s + 1} \quad (5.46)$$

where:

ξ = *damping factor (or damping ratio)*

The form of the Inverse Laplace Transform of this function depends on whether the Damping factor ξ is less than, equal to, or greater than one. The Inverse Laplace Transform is relatively complex and is not shown here.

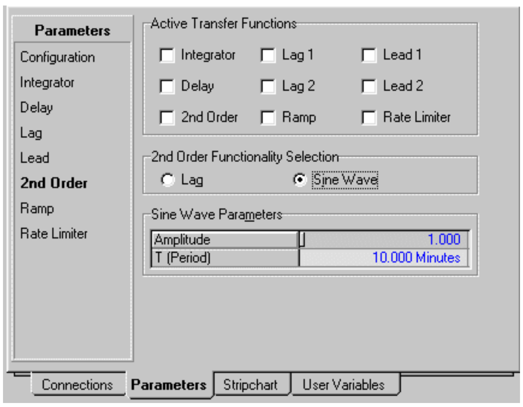
A standard second-order Lead or Lag transfer function may or may not produce an oscillatory output, depending on the damping factor ξ ("Xi"). If the damping factor is unity, the response is said to be critically damped. If $\xi > 1$, the process is overdamped, producing a slower response than the critically damped case. If $\xi < 1$, the process is underdamped, producing the faster response. However, the response overshoots the target value, and oscillates with a period *T*.

Select the **2nd Order** checkbox to simulate the Second Order Process.

First Order, Delay, and Ramp Functions can also be active, in which case all equations are superimposed.

Sine Wave

Figure 5.193



The Sine Wave Transfer function is defined as follows:

$$G = \frac{K\omega}{s^2 + \omega^2} \tag{5.47}$$

where:

ω = *frequency of oscillation*

K = amplitude

The frequency is the inverse of the period ($\omega = 1/T$).

The Inverse Laplace of the Sine Wave Transfer Function is:

$$f(t) = K \sin \omega t \tag{5.48}$$

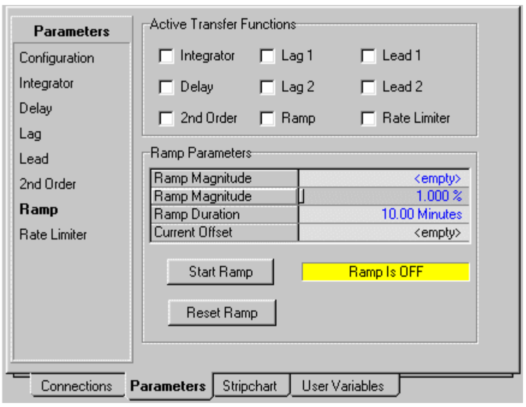
The K-value (transfer function gain) is the amplitude of the sine wave, in a percentage of the Signal range. The range is the difference between the Signal Minimum and Maximum values given on the Configuration page.

As usual, select the Sine Wave radio button on the 2nd Order page to simulate the Sine Wave. You cannot activate both the Standard 2nd Order and the Sine Wave at the same time.

Ramp Page

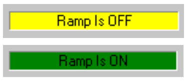
The Ramp Page allows you to ramp the output value (OP) linearly in a transfer function.

Figure 5.194



The Ramp Parameters group contains the following parameters:

- **Ramp Magnitude**. Allows you to specify the amount of ramp either as a magnitude in the OP units, or as a percentage of the OP range. A positive ramp magnitude represents an increase in signal, whereas a negative value represents a decrease in signal.
- **Ramp Duration**. Allows you to specify the total amount of time required for the ramp function to change the OP.
- **Current Offset**. Displays the amount of deviation between the original OP, and the ramped OP.



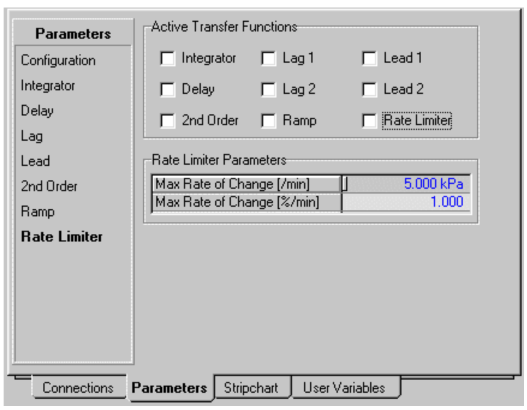
Ramp status bar

You can execute the Ramp by clicking on the Start Ramp button. The status of the ramp is shown in the ramp status bar. Once the ramp is running, the Start Ramp button automatically changes to a Stop Ramp button. You can stop the ramp at any given time by clicking on the Stop Ramp button. As the ramp is executing, the Ramp Magnitude, and Ramp Duration start approaching to zero. When the Ramp Duration reaches zero, the ramp stops. You can reset the OP to the original value (before ramped) by clicking on the Reset Ramp button.

Rate Limiter Page

The Rate Limiter page allows you to specify the maximum rate of change of the OP.

Figure 5.195



The Rate Limiter analyzes the signal transformation in the transfer function, and limits the OP to change by a user-specified maximum. The OP is restricted to change faster than the preset maximum. Therefore, any abrupt changes in the input signals can be intercepted, and smoothed.

To set the maximum rate of change of the OP, you can specify one of the following parameters in the Rate Limiter Parameters group:

- **Max Rate of Change (/min)**. Allows you to specify the magnitude of the maximum rate of change in the OP units.
- **Max Rate of Change (%/min)**. Allows you to specify the percentage of the maximum rate of change in the OP range.

5.12.4 Stripchart Tab

Refer to **Section 1.3.7 - Stripchart Page/Tab** for more information.

The Stripchart tab allows you to select and create default strip charts containing various variable associated to the operation.

5.12.5 User Variables Tab

For more information refer to **Section 1.3.8 - User Variables Page/Tab**.

The User Variables tab enables you to create and implement your own user variables for the current operation.

5.13 Common Options

5.13.1 ATV Tuning Technique

The ATV (Auto Tune Variation) Technique is one of a number of techniques used to determine two important system constants known as the Ultimate Period, and the Ultimate Gain. From these constants, tuning values for proportional, integral, and derivative gains can be determined.

The Tuning option only sets up the limit cycle; it does not calculate the tuning parameters for you.

A small limit-cycle disturbance is set up between the Control Output and the Controlled Variable, such that whenever the process variable crosses the set point, the controller output is changed. The ATV Tuning Method is as follows:

- Determine a reasonable value for the valve change (OP). Let *h* represent this value. In HYSYS, *h* is 5%.
- Move the valve +*h*%.
- Wait until the process variable starts moving, then move valve -2*h*%.
- When the PV crosses the set point, move the OP +2*h*%.

To set up a Strip Chart to track the PV and OP do the following:

1. To open the Databook property view, press **CTRL D**.
2. On the Variables tab, add the PV and OP to the Variable List.
3. On the Strip Charts tab, add a Strip Chart and activate the PV and OP.
4. View the Strip Chart.

- Continue this procedure until the limit-cycle is established.

From the cycle, two key parameters can be observed:

Observed Parameter	Description
Amplitude (*a*)	The amplitude of the PV curve, as a fraction of the PV span.
Ultimate Period (*PU*)	Peak-to-Peak period of the PV curve.

The Ultimate Gain can be calculated from the following relationship:

$$KU = \frac{4h}{\pi a} \tag{5.49}$$

where:

KU = ultimate gain

h = change in OP (0.05)

a = amplitude

Finally, the Controller Gain and Integral Time can be calculated as follows:

Controller Gain = KU / 3.2

*Controller Integral Time = 2.2*PU*

The ATV Tuning Method only works for systems with dead time.

5.13.2 Controller Face Plate

Refer to **Section 11.8 - Face Plates** in the **HYSYS User Guide** for more information.

There are two ways that you can access the controller Face Plate:

1. In the menu bar select **Tools | Face Plates** command, or press **CTRL F**.

   The Face Plates property view appears.
2. From the list of available flowsheets, select the flowsheet that contains the logical operation you want to view the face plate for.
3. From the list of available logical operations, select the logical operation you want to view the face plate for.
4. Click the **Open** button. The Face Plates property view closes and the face plate for the selected logical operation appears.

OR

1. Open the property view of a Controller operation.
2. Click the **Face Plate** button located at the bottom of the Controller’s property view.

Each controller’s Face Plate varies in appearance, however the functionality remains the same. This section provides a general description of how to use the controller Face Plate.

The Face Plate provides all pertinent information about the controller when the simulation is running. The Setpoint is shown as a red pointer, and the actual value of the Process Variable appears in the current default unit. Output is always displayed as a percentage of the span you defined on the Valve tab. The Face Plate also displays the execution type and the setpoint source.

Also, you can change the mode of the Controller by selecting the mode from the drop-down list at the bottom left of the Face Plate. The mode choices are identical to those on the Parameters tab. Clicking the Tuning button returns you to the Tuning tab of the Controller property view.

Figure 5.196

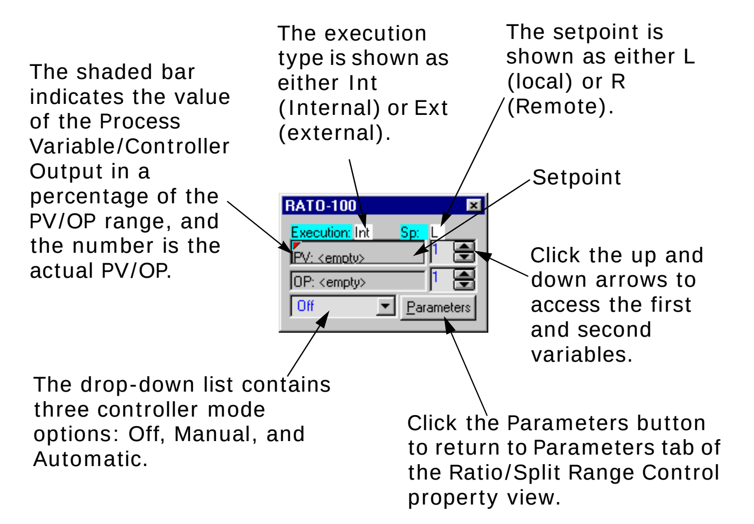
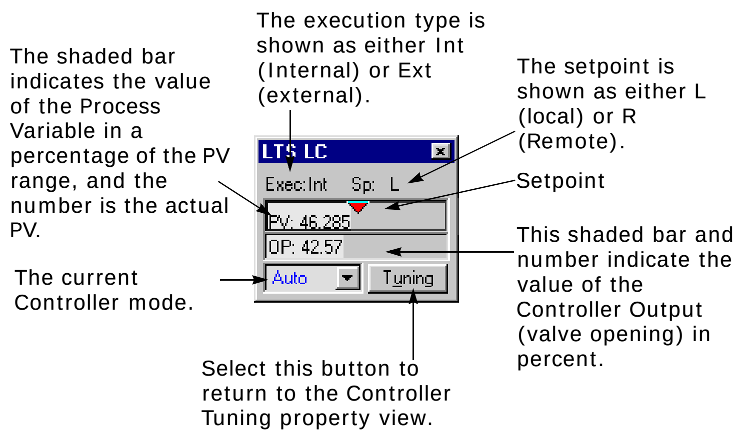


Figure 5.197



Changing the Setpoint and Output

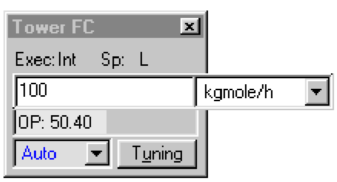
You can change the SP or OP of the Controller (depending on the current mode) at any time during the simulation without returning to the Parameters tab, by using the Face Plate.

To change the SP while in Automatic mode, or to change the OP while in Manual mode, use any one of the following three methods:

1. Move to the field for the parameter you want to change.

   For this example, the Setpoint (top field) is changed. Start entering a new value for the SP, and HYSYS displays a field with a drop-down list containing the default units. Once you have entered the value, press **ENTER** and HYSYS accepts the new Setpoint.

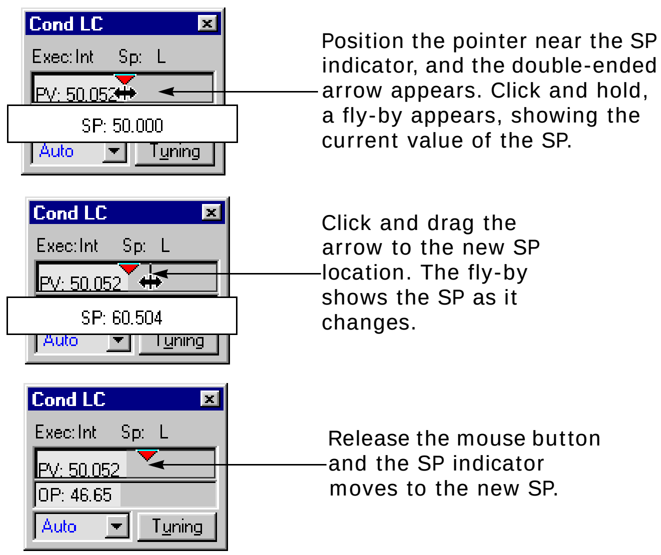
Figure 5.198



If you select an alternate unit, your value appears in the face plate using HYSYS display units.

2. Place the mouse pointer near the red Setpoint indicator, and the cursor changes to a double-ended arrow. Click and hold, a fly-by appears below, showing the current value of the SP (in this case, 50%).
3. Click and drag the double-ended arrow to the new SP of 60%. The fly-by displays the SP value as you drag. Release the mouse button to accept the new SP.

Figure 5.199



4. Place the pointer at either end of the field, and the pointer changes to a single-ended arrow. Click once to increase or decrease the value by 1%. For example, switch to Manual mode and adjust the OP. To increase the OP, move the pointer to the right end of the field and the single-ended arrow pointing to the right appears. Click to increase the OP by 1%.

You can click the button consecutively to repeatedly increase (or decrease) the OP.

Object Inspect Menu of Face Plates

The Object Inspect menu for a Fixed Size Face Plate is shown in the figure below.

Figure 5.200

The options associated with this menu are:

Command	Description
Turn Off	Turns the Controller Mode to Off.
Start Ramp	Starts Setpoint Ramping
Auto-Tune	Puts the Controller into a cycling mode. This can be used for tuning the Controller.
Tuning	Returns you to the Tuning page of the Controller property view.
Connections	Returns you to the Connections page of the Controller property view.
Parameters	Returns you to the Parameters page of the Controller property view.
Print Datasheet	Allows you to print the Datasheet for the controller.
Print Specsheet	Allows you to print the controller Specsheet.

Refer to **Chapter 3 - Control Theory** in the **HYSYS Dynamic Modeling** guide for information on the object inspection options.

The additional menu options in the Object Inspection menu for a Scalable Face Plate are:

Command	Description
Font	Allows you to choose the Font for the text on the Face Plate.
Hide Values/ Show Values	Hides the values for SP, PV, and OP. When the values are hidden, the Show Values option appears. Choose this to display the values.
Hide Units/Show Units	Hides the units for SP and PV. When the units are hidden, the Show Units option appears in the menu. Choose this to display the units.

6 Optimizer Operation

6.1 Optimizer

HYSYS contains a multi-variable steady state Optimizer. Once your flowsheet has been built and a converged solution has been obtained, you can use the Optimizer to find the operating conditions which minimize (or maximize) an Objective Function. The object-oriented design of HYSYS makes the Optimizer extremely powerful, since it has access to a wide range of process variables for your optimization study.

The Optimizer is available for steady state calculations only. The operation does not run in Dynamic mode.

The Optimizer owns its own Spreadsheet for defining the Objective Function, as well as any constraint expressions to be used. The flexibility of this approach allows you, for example, to construct Objective Functions which maximize profit, minimize utilities or minimize Exchanger UA.

The following terminology is used in describing the Optimizer.

Terms	Definition
Primary Variables	These are the variables imported from the flowsheet whose values are manipulated in order to minimize (or maximize) the objective function. You set the upper and lower bounds for all of the primary variables, which are used to set the search range, as well as for normalization.
Objective Function	The function which is to be minimized or maximized. There is a great deal of flexibility in describing the Objective Function; primary variables can be imported and functions defined within the Optimizer Spreadsheet, which possesses the full capabilities of the main flowsheet spreadsheet.
Constraint Functions	Inequality and Equality Constraint functions can be defined in the Optimizer Spreadsheet. An example of a constraint is the product of two variables satisfying an inequality (for example, -A*B<K). The BOX, Mixed, and Sequential Quadratic Programming (SQP) methods are available for constrained minimization with inequality constraints. Only the Original and Hyprotech SQP methods can handle equality constraints. The Fletcher-Reeves and Quasi-Newton methods are available for unconstrained optimization problems.

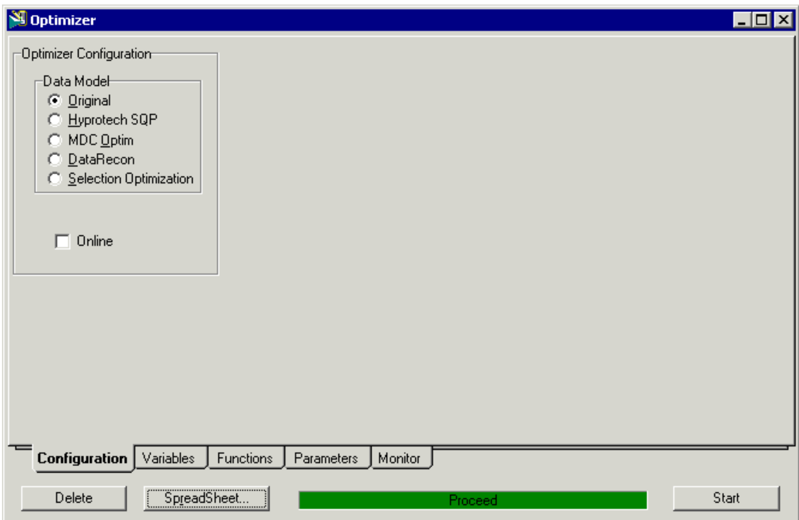
You have the ability to define not only how the Optimizer Function is set up, but also how the Optimizer reaches a solution. You can set parameters such as the Optimization Scheme used, the Maximum Number of Iterations, and the Tolerance.

6.1.1 General Optimizer Property View

To open the Optimizer, select **Simulation | Optimizer** command from the menu bar, or press **F5**.

When you first open the Optimizer, the figure below appears.

Figure 6.1



The amount of tabs in the Optimizer property view changes depending on which mode of Optimizer you select. The first tab, called the Configuration tab, remains no matter which mode of Optimizer you select.

Three buttons are available on the Optimizer property view, no matter which tab is being viewed, or which mode of Optimizer has been selected.

Buttons	Description
Delete	Erases all the current information from the Optimizer and its Spreadsheet.
Spreadsheet	Accesses the Optimizer's dedicated Spreadsheet.
Start/Stop	Starts or stops the Optimizer calculations. An objective function must be defined prior to the start of the calculations.

6.1.2 Configuration Tab

The Configuration tab allows you to select the Optimizer mode you want, by selecting appropriate radio button in the Data Model group.

The Configuration tab is the same no matter which Optimizer mode you select.

HYSYS has five modes of Optimizer:

Refer to **Section 6.2 - Original Optimizer** for more information.

Refer to **Section 6.3 - Hyprotech SQP Optimizer** for more information.

Refer to **Section 6.4 - Selection Optimization** for more information.

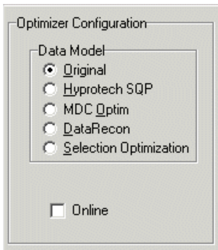
- **Original**. The Default option from HYSYS 2.4.
- **Hyprotech SQP**. The new Optimizer available for HYSYS 3.0.
- **MDC Optim**. The Optimization option from HYSYS 2.4. Refer to **Chapter 3 - Optimizer** in the **Aspen RTO Reference Guide** for more information.
- **DataRecon**. The DataRecon option from HYSYS 2.4. Refer to **Chapter 5 - DRU Overview** in the **Aspen RTO Reference Guide** for more information.
- **Selection Optimization**. The Selection Optimization option available for HYSYS 3.1.

6.2 Original Optimizer

To access the Original Optimizer:

1. On the **Configuration** tab, select the **Original** radio button in the Data Model group, as shown in the figure below.

Figure 6.2



2. The Original Optimizer property view contains the following tabs:

Refer to **Section 6.1.2 - Configuration Tab** for more information.

- Configuration

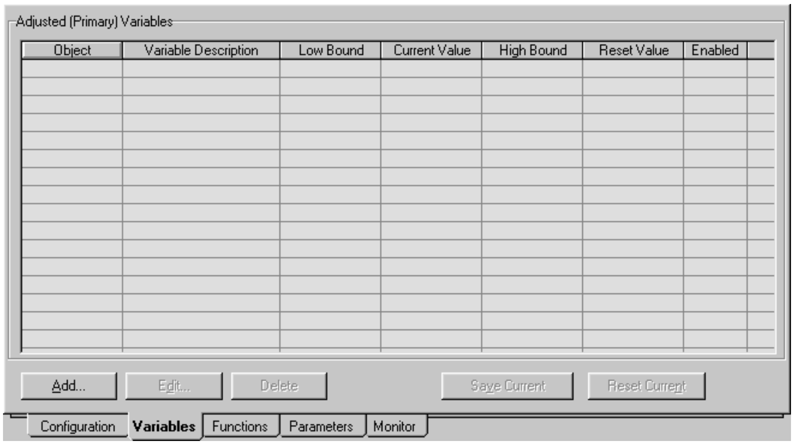
Configuration tab is the same for all Optimizer mode.

- Variable
- Functions
- Parameters
- Monitor

6.2.1 Variables Tab

When you invoke the Optimizer for the first time, the Variables tab appears as shown in the figure below:

Figure 6.3



The Variables tab is only available if you select the Original configuration.

On the Variables tab, you can import the primary variables which minimize or maximize the objective function. Any process variable that is modifiable (user-specified) can be used as a primary variable. New variables are added via the Variable Navigator. The five buttons at the bottom of the table allow you to manipulate the variables.

Refer to **Section 1.3.9 - Variable Navigator Property View** for information on the Variable Navigator.

Button	Description
Add	Allows you to add the primary variables. When you click this button the Variable Navigator property view appears, and you can select the variable you want from the list on the property view. Only user-specified variables can be used as Primary Variables.
Edit	Allows you to edit the selected primary variables for the variable you want to change.
Delete	Allows you to the remove the selected variable.

Button	Description
Save Current	Stores the current value as the Reset value.
Reset Current	Resets current values to the Reset value.

All variables must be given upper and lower bounds, which are used to normalize the Primary Variable:

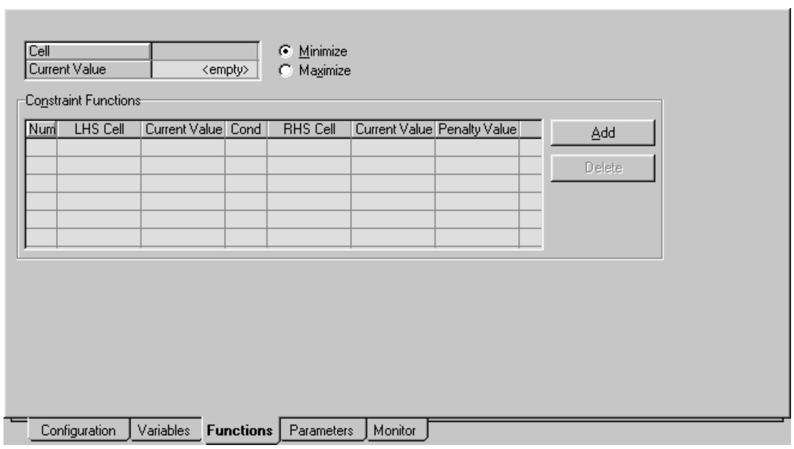
$$x_{norm} = \frac{x - x_{low}}{x_{high} - x_{low}} \tag{6.1}$$

The upper and lower bound for each Primary Variable should be chosen such that a reasonable flowsheet solution is obtained within the entire range. For example, assume that the Primary Variable is the Molar Flow of a stream being fed to the tube side of a heat exchanger. If this Molar Flow is too low, a temperature cross may result in the heat exchanger, which stops the Optimizer calculations. In this case, the lower bound should be chosen such that the temperature cross does not occur.

6.2.2 Functions Tab

The Functions tab contains two fields, two radio buttons and the Constraints Functions group.

Figure 6.4



The Functions tab is only available if you select the Original configuration.

For information on using the Spreadsheet, refer to **Chapter 5 - Logical Operations**.

The Optimizer possesses a dedicated Spreadsheet which is used to develop the Objective function, as well as any Constraint functions to be used.

To open the Optimizer Spreadsheet, click the SpreadSheet button.

The Optimizer's Spreadsheet is identical to the Spreadsheet operation; process variables can be attached by dragging and dropping, or using the Variable Navigator. Once the necessary process variables are connected to the Spreadsheet, you can construct the Objective Function and any constraints using the standard syntax.

You can specify the Objective Function in the Cell field. The current value of the objective function is provided in the display field below the Cell field. Further, the objective function group is the location where you can specify (via radio buttons) to minimize or maximize the objective function.

The Constraint Functions group is where you can specify the left and right sides of the Constraint function (in the LHS Cell and RHS Cell columns). Specify the relationship between the left hand and right hand cell (LHS > RHS, LHS < RHS, LHS = RHS) in the Cond column. The Constraint Function is multiplied by the Penalty Value in the Optimization calculations. If you find that a constraint is not being met, increase the Penalty Value; the higher the Penalty Value, the more weight that is given to that constraint. The Penalty Value is equal to 1 by default.

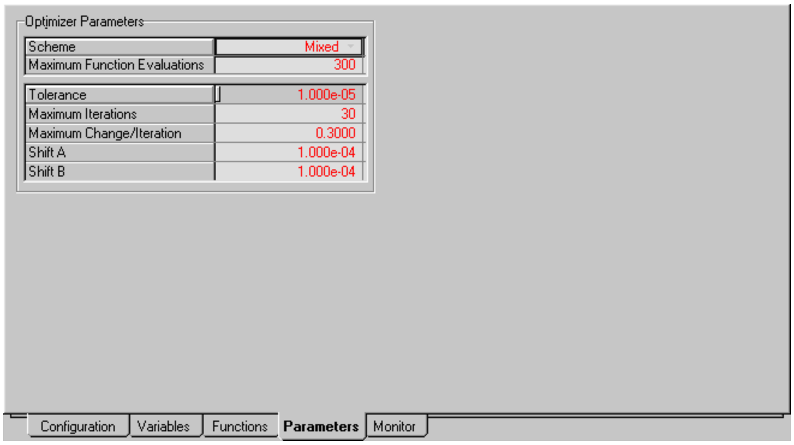
The BOX, Mixed, and SQP Methods allow for Inequality Constraints. Only the SQP Method incorporates Equality Constraints.

The current values of the Objective Function and the left and right sides of the Constraint Function cells appear in their respective fields.

6.2.3 Parameters Tab

The Parameters tab is used for selecting the Optimization Scheme and defining associated parameters.

Figure 6.5



The Parameters tab is only available if you select the Original configuration.

The following table contains a description of each parameter available.

Refer to **Section 6.2.5 - Optimization Schemes** for more information about the schemes.

Parameters	Description
Scheme	You can select the scheme type from the drop-down list.
Maximum Function Evaluation	Sets the maximum number of function evaluations (not to be confused with the maximum number of iterations). During each iteration, the relevant portion of the flowsheet is solved several times, depending on factors such as the Optimization Scheme, and number of primary variables. Primary Variables are normalized. $x_{norm} = \frac{x - x_{low}}{x_{high} - x_{low}}$
Tolerance	HYSYS determines the change in the objective function between iterations, as well as the changes in the normalized primary variables. Using this information, HYSYS determines if the specified tolerance is met.

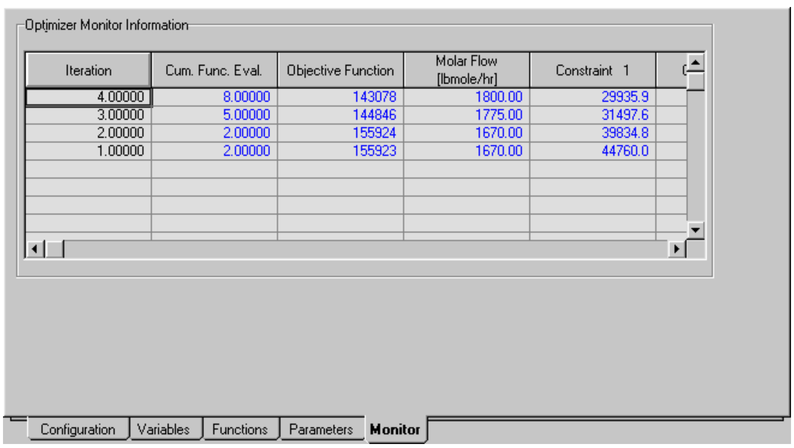
Parameters	Description
Maximum Iteration	The maximum number of iterations. Calculations stop if the maximum number of iterations is reached. All of the methods except the BOX method use derivatives.
Maximum Change / Iteration	The maximum allowable change in the normalized primary variables between iterations. For instance, assume the maximum change per iteration is 0.3 (this is the default value). If you have specified molar flow as a primary variable with range 0 to 200 kgmole/hr, then the maximum change in one iteration would be (200)(0.3) or 60 kgmole/hr. Shift B ensures that the Shift interval x_{Shift} never be zero.
Shift A / Shift B	Derivatives of the objective function and/or constraint functions with respect to the primary variables are generally required and are calculated using numerical differentiation. The numerical derivative is calculated from the following relationship: $$x_{shift} = ShiftA * x + ShiftB$$ where: *x = perturbed variable (normalized)* *x_{shift} = shift interval (normalized)* Derivatives are calculated using: $$\frac{\partial y}{\partial x} = \frac{y_2 - y_1}{x_{shift}}$$ *where:* *y2 = value of the affected variable corresponding to x + x_{shift}* *y1 = value of the affected variable corresponding to x* Prior to each step, the Optimizer needs to determine the gradient of the optimization surface at the current location. The Optimizer moves each primary variable by a value of x_{shift} (which due to the size of Shift A and Shift B be a very small step). The derivative is then evaluated for every function (Objective and Constraint) using the values for y at the two locations of x. From this information and the Optimizer history, the next step direction and size are chosen. In general, it should not be necessary to change Shift A and Shift B from their defaults. Some Schemes move all Primary variables simultaneously, while others move them sequentially.

Parameters	Description
	To determine each derivative, a variable evaluation must be made in addition to the main flowsheet evaluation which is done after each iteration (main step change). Therefore, if there are two primary variables, there are three function evaluations for every iteration. If you have selected the Mixed Optimizer Scheme, the BOX and SQP methods are used in sequence - this is the reason why the Function Evaluations are reset part way through the calculations.

6.2.4 Monitor Tab

The Monitor tab displays the values of the objective function, primary variables, and constraint functions during the Optimizer calculations. New information is updated only when there is an improvement in the value of the Objective Function. The constraint values are positive if inequality constraints are satisfied and negative if inequality constraints are not satisfied.

Figure 6.6



The Monitor tab is only available if you select the Original configuration.

6.2.5 Optimization Schemes

The following sections describe the Optimization schemes for the Original Optimizer.

Function Setup

The Optimizer manipulates the values of a set of primary variables in order to minimize (or maximize) a user-defined objective function, constructed from any number of process variables.

$$\min f(x_1,\ x_2,\ x_3,\ ...,\ x_n) \tag{6.2}$$

where:

$x_1, x_2, ..., x_n$ = *process variables*

In general, the primary variables should not be part of the Objective Function.

Each primary variable, x^0, can be manipulated within a specified range:

$$x^0_{i\ LowerBound} < x^0_i < x^0_{i\ UpperBound} \quad \text{with}\ \ i = 1,\ ...,\ j \tag{6.3}$$

where:

x_i = *a process variable used to define the Objective Function*

x_i^0 = *a primary variable which is manipulated by the Optimizer*

y_i = *a variable used to define the Constraint Function*

The general equality and inequality constraints are:

$$\begin{array}{ll} c_i(y_1,\ y_2,\ y_3,\ \ldots,\ y_n) = 0, & i\ =\ 1,\ \ldots,\ m_1 \\ c_i(y_1,\ y_2,\ y_3,\ \ldots,\ y_n) \le 0, \quad with & i\ =\ m_1+1,\ \ldots,\ m_2 \\ c_i(y_1,\ y_2,\ y_3,\ \ldots,\ y_n) \ge 0, & i\ =\ m_2+1,\ \ldots,\ m \end{array} \tag{6.4}$$

The constraint functions should generally not use the primary variables.

All primary variables are normalized from the lower bound through the upper bound. Thus, reasonable lower and upper bounds must be specified. Exceedingly high or low variable bounds should obviously be avoided as they may result in numerical problems when scaling. An initial starting point must be specified, and it should be within the feasible region. Constraints are optional and are not supported by all of the Optimization Schemes.

Refer to **Section 11.7 - Databook** in the **HYSYS User Guide**.

HYSYS recommends users to manually manipulate the primary variables to get a feel for the appropriate boundaries. Use the Data Recorder or Case Study tool for this purpose.

If HYSYS fails to evaluate the objective function or any of the constraint functions, the Optimizer reduces the incremental step of the last primary variable by a half. The flowsheet is then recalculated. If the function evaluation is still unsuccessful, the optimization stops.

By default, the Optimizer is set up to minimize the objective function. A Maximize radio button is provided on the Functions tab if you want to maximize an objective function. Internally the Optimizer simply reverses the sign.

BOX Method

The procedure is loosely based on the “Complex” method of BOX[1]; the Downhill Simplex algorithm of Press et al[2] and the BOX algorithm of Kuester and Mize.[3]

The BOX Method only handles inequality constraints.

The BOX method is a sequential search technique which solves problems with non-linear objective functions, subject to non-linear inequality constraints. No derivatives are required. It handles inequality constraints but not equality constraints. The BOX method is not very efficient in terms of the required number of function evaluations. It generally requires a large number of iterations to converge on the solution. However, if applicable, this method can be very robust.

Procedure:

1. Given a feasible starting point, the program generates an original “complex” of n+1 points around the centre of the feasible region (where n is the number of variables).
2. The objective function is evaluated at each point. The point having the highest function value is replaced by a point obtained by extrapolating through the face of the complex across from the high point (reflection).
3. If the new point is successful in reducing the objective function, HYSYS tries an additional extrapolation. Otherwise, if the new point is worse than the second highest point, HYSYS does a one-dimensional contraction.
4. If a point persists in giving high values, all points are contracted around the lowest point.
5. The new point must satisfy both the variable bounds and the inequality constraints. If it violated the bounds, it is brought to the bound. If it violated the constraints, the point is moved progressively towards the centroid of the remaining points until the constraints are satisfied.
6. Steps #2 through #5 are repeated until convergence.

SQP Method

The Sequential Quadratic Programming (SQP) Method handles inequality and equality constraints.

SQP is considered by many to be the most efficient method for minimization with general linear and non-linear constraints, provided a reasonable initial point is used and the number of primary variables is small.

The implemented procedure is based entirely on the Harwell subroutines VF13 and VE17[4]. The program follows closely the algorithm of Powell[5].

It minimizes a quadratic approximation of the Lagrangian function subjected to linear approximations of the constraints. The second derivative matrix of the Lagrangian function is estimated automatically. A line search procedure utilizing the “watchdog” technique (Chamberlain and Powell[6]) is used to force convergence.

Mixed Method

The Mixed method attempts to take advantage of the global convergence characteristics of the BOX method and the efficiency of the SQP method. It starts the minimization with the BOX method using a very loose convergence tolerance (50 times the desired tolerance). After convergence, the SQP method is then used to locate the final solution using the desired tolerance.

The Mixed Method handles inequality constraints only.

Fletcher Reeves Method

The procedure implemented is the Polak-Ribiere modification of the Fletcher-Reeves conjugate gradient scheme. The approach closely follows that of Press et al[2], with modifications to allow for lower and upper variable bounds. This method is efficient for general minimization with no constraints.

The Fletcher Reeves (Conjugate Gradient) Method does not handle constraints.

The method used for the one-dimensional search can be found in reference 2, listed at the end of this chapter.

Procedure:

1. Given a starting point evaluate the derivatives of the objective function with respect to the primary variables.
2. Evaluate the new search direction as the conjugate to the old gradient.
3. Perform one-dimensional search along the new direction until the local minimum has been reached.
4. If any variable exceeds its bound, bring it back to the bound.
5. Repeat steps #1 through #4 until convergence.

Quasi-Newton Method

The Quasi-Newton method of Broyden-Fletcher-Goldfarb-Shanno (BFGS) according to Press et al[2] has been implemented. In terms of applicability and limitations, this method is similar to the of Fletcher-Reeves method.

The Quasi-Newton Method does not handle constraints.

The Quasi-Newton method calculates the new search directions from approximations of the inverse of the Hessian Matrix.

Method	Unconstrained Problems	Constrained Problems: Inequality	Constrained Problems: Equality	Calculates Derivatives
BOX	X	X		
Mixed	X	X		X
SQP	X	X	X	X
Fletcher-Reeves	X			X
Quasi-Newton	X			X

6.2.6 Optimizer Tips

The following are setup tips for the Original Optimizer.

1. Reasonable upper and lower variable bounds are extremely important. This is necessary not only to prevent bad flowsheet conditions (for example temperature crossovers in Heat Exchangers), but also because variables are scaled between zero and one in the optimization algorithms using these bounds.
2. For the **BOX** and **Mixed** methods, the Maximum Change/Iteration of the primary variables (set on the **Parameters** tab) should be reduced. A value of 0.05 or 0.1 is more appropriate.
3. The **Mixed** method generally requires the least number of function evaluations (in other words, is the most efficient).
4. If the **BOX**, **Mixed** or **SQP** Methods are not honouring your constraints, try increasing the Penalty Value on the **Functions** tab by 3 or 6 orders of magnitude (up to a value similar to the expected value of the objective function). In other words, it is helpful to attempt to get the magnitude of the objective function and penalty as similar as possible (especially when the BOX Method is used).
5. By default the Optimizer minimizes the objective function. You can maximize the objective function by selecting the **Maximize** radio button on the **Functions** tab.

Internally, the Optimizer multiplies the objective function by minus one for maximization.

6.3 Hyprotech SQP Optimizer

The Hyprotech SQP is a sequential quadratic programming (SQP) algorithm incorporating an L1-merit function and a BFGS approximation to the Hessian of the Lagrangian. The algorithm features step size restriction, decision variable and objective function scaling, a basic watchdog method, and a problem-independent and scale-independent relative convergence test. The algorithm also ensures that the model is evaluated only at points feasible with respect to the variable bounds.

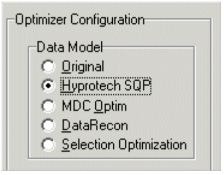
Refer to the **Aspen RTO Reference Guide** for details.

The Hyprotech SQP requires the use of Derivative Utilities.

To access the Hyprotech SQP Optimizer:

1. On the **Configuration** tab, select the **Hyprotech SQP** radio button in the Data Model group, as shown in the figure below:

Figure 6.7



2. The Hyprotech SQP Optimizer property view contains two tabs:
   - Configuration

Refer to **Section 6.1.2 - Configuration Tab** for more information.

Configuration tab is the same for all Optimizer mode.

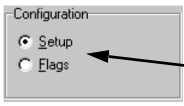
   - Hyprotech SQP

6.3.1 Hyprotech SQP Tab

The Hyprotech SQP tab allows you to manipulate the configurations setup and flags.

The Hyprotech SQP tab is only available if you select Hyprotech SQP configuration.

Figure 6.8

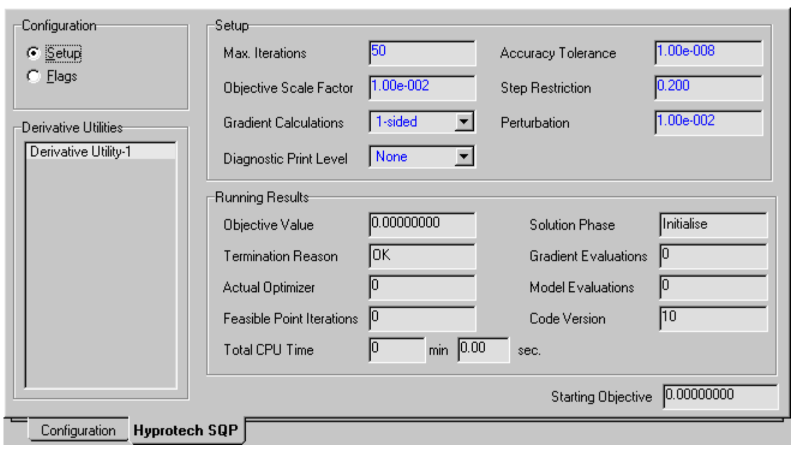


To access these options, select the corresponding radio buttons in the Configuration group.

Setup Option

If you select the Setup radio button from the Configuration group, the Hyprotech SQP tab appears as shown in the figure below.

Figure 6.9



The Starting Objective display field at the bottom of the tab gives the objective function value at the starting point, before carrying out any optimization. The value is unscaled and you cannot change the value.

The parameters available in the Setup option are sorted into two groups:

- Setup
- Running Results

Setup Group

You can change the values of the parameters in the Setup group. The group contains the following parameters:

Variables	Description
Max. Iterations	The maximum number of major iterations. A major iteration consists of a sequence of minor iterations that minimize a linearly constrained sub-problem.
Objective Scale Factor	Used for scaling the objective function. Positive values are used as-is, negative values use the factor abs(scale*F) (where F is the initial objective function value) and a value of 0.0 a factor is generated automatically.
Gradient Calculations	Specifies if one-sided (forward) or two-sided (central) differences are to be used for gradient calculations. In both cases, the perturbation size used for the Optimizer internal variables is given by the Perturbation property.
Diagnostic Print Level	Selects the amount of information to include in the Optimizer diagnostic file.
Accuracy Tolerance	A relative accuracy tolerance used in the test for convergence. The following convergence test is used, $$\text{ConvergenceSum} \le \text{OptimalityTolerance} \times \max(\lvert F(x)\rvert,\ 1.0)$$ *where:* $$\text{ConvergenceSum} = \lvert \nabla F(x)^r d \rvert + \sum_{j=1}^{M} \lvert u_j C_j(x) \rvert$$ The ConvergenceSum is a weighted sum of possible objective function improvement and constraint violations, and has the same units as the objective function. This allows the same tolerance parameter to be used for different problems, and makes the convergence test independent of objective function scaling.

Variables	Description
Step Restriction	A line search step-size restriction factor used during the first 3 iterations. Values greater than 1.0 result in no step restriction. Set the factor to 1.0, 10^{-1}, 10^{-2}, and so forth, to impose larger restrictions.
Perturbation	The change in size of the scaled variables is used in gradient evaluation. Individual variables are scaled according to the variable Minimum and Maximum properties (or the Range property if the **Fix Variable Spans property** checkbox is selected).

Running Results Group

You cannot change the values of the parameters in the Running Results group. The group contains the following parameters:

Variables	Description
Objective Value	Displays the current plant model objective function value as calculated by the Optimizer.
Termination Reason	Displays the termination status of the Optimizer. Values include Running, Step convergence, Unbounded, Impossible, Not run, and Stopped.
Actual Optimizer	Displays the number of major iterations.
Feasible Point Iterations	Displays the number of minor iterations since the last major iteration.
Total CPU Time	Reports the time taken to solve the optimization problem.
Solution Phase	Displays the current phase of the Optimizer algorithm. Values include Initialize, Setup, OPT Deriv, OPT Search, and Results.
Gradient Evaluations	Reports the number of gradient evaluations performed during the course of the optimization.
Model Evaluations	Reports the number of model evaluations performed during the course of the optimization.
Code Version	The version of Optimizer.

Results

The results produced at the end of the optimization run are as follows:

- Values of the Optimizer constraints, variables, and objective function.
- Shadow prices for active constraints.
- A termination reason.

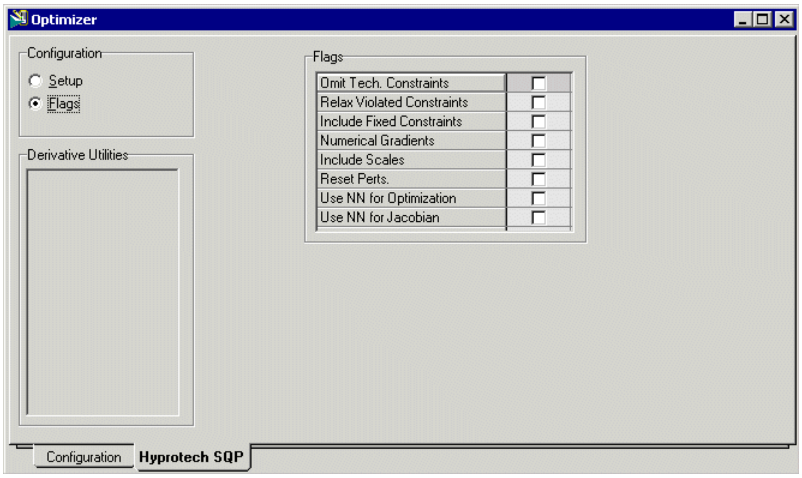
- Iterations and CPU time taken.

Some of the above mentioned results can be seen in the Running Results group, and the other results are found in the Derivative utilities.

Flags Option

If you select the Flags radio button from the Configuration group, the Hyprotech SQP tab appears as shown in the figure below.

Figure 6.10



The following table explains the options available in Flags group:

Options	Description
Omit. Tech Constraints	Not used.
Relax Violated Constraints	Not used.
Include Fixed Constraints	If this checkbox is selected then the Optimizer variables that have their **Optimize Flag property** checkbox selected are included in the optimization even if they have equal Minimum and Maximum values.
Numerical Gradients	Not used.

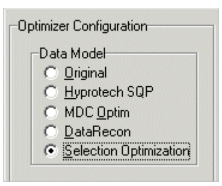
Options	Description
Include Scales	Includes the variable Range properties as scaling factors within the algorithm.
Reset Perts.	Used at the start of optimization to indicate that the gradient calculation process removes noise elements (activated) or not (deactivated).
Use NN for Optimization	Allows you to use any trained Neural Network in the flowsheet to replace the traditional HYSYS solver for optimization. This improves the robustness of the model, and reduces the calculation time thereby improving overall performance. However, the accuracy in the solution depends upon how the NN is trained and the data available. Upon solving the NN's are unembedded and the flowsheet solves at the optimizer given values.
Use NN for Jacobian	Allows you to use any trained Neural Network to calculate the Jacobian. This is used to determine the next step in the Optimization process. This is slower than the above option but is more accurate. See Optimization above for more details.

Refer to **Section 14.14 - Parametric Utility** for more details on NN's.

6.4 Selection Optimization

The Selection Optimization consists of algorithms that solve Mixed Integer Non-Linear Programming (MINLP) problems, in which the objective function is minimized by adjusting both the real-valued decision variables, and binary-valued decision variables. These binary, or discrete state variables can be used to represent the state of the equipment (On, Off, Out of Service, and Always in Service) in the Derivative Utility. The algorithms attempts to select a combination of discrete states that both satisfy the constraints, and minimize the objective function. There are two MINLP methods available: Stochastic (also known as the simulated annealing method), and Branch and Bound. These methods use Non-Linear Programming (NLP) optimizers (Hyprotech SQP, and MDC Optim) to solve sub-problems.

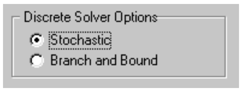
To access the Selection Optimization:



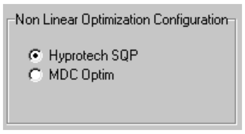
1. On the **Configuration** tab, select the **Selection Optimization** radio button in the Data Model group.
2. Click on the **Selection Optimization** tab.

6.4.1 Selection Optimization Tab

The Selection Optimization tab allows you to select, and configure the type of discrete solver, and non-linear optimizer. Two base groups are available on the Selection Optimization tab:



- **Discrete Solver Options**. You can select the Stochastic, or Branch and Bound solver by clicking on the appropriate radio button. Depending on the type of discrete solver you selected, additional groups appear on the Selection Optimization tab.



- **Non Linear Optimization Configuration**. There are two non- linear optimizers available: Hyprotech SQP, and MDC Optim. You can select the type of non-linear optimizer by clicking on the appropriate radio button. Depending on the type of non-linear optimizer you selected, additional tabs appear on the Optimizer property view for further configuration.

For more information, refer to Hyprotech SQP Optimizer (**Section 6.3 - Hyprotech SQP Optimizer**), and MDC Optimizer (**Chapter 5 - DRU Overview** in the **Aspen RTO Reference Guide**).

The following sections describe the Stochastic, and Branch and Bound discrete solving methods.

Stochastic Method

The Stochastic method is a simulated annealing algorithm, which is derived from the statistical mechanics for finding near globally optimum solutions in non-linear integer problems.

The algorithm is based on the analogy between the annealing of solids, and combinatorial optimization. The analogies are as follows:

- The states of the solids in annealing represent the feasible solutions of the optimization problem.
- The energies of state in annealing correspond to the value of the objective function.
- The minimum energy state in annealing corresponds to the optimum solution.
- Rapid quenching (fast cooling) corresponds to local optimum in the optimization problem.

At a given temperature, the probability distribution of the system energies is determined by the Boltzmann function:

$$P(E) \propto e^{-E/(kT)} \qquad (6.5)$$

where:

E = System energy

k = Boltzmann constant

T = Temperature

P(E) = Probability of the system in a state with E energy

The simulated annealing algorithm implemented in the Stochastic method uses a criterion similar to the Boltzmann probability function. The criterion states that if the difference between the objective function values of the current and the newly produced solution is equal to or larger than zero, a random number, ∂,with uniform distribution [0,1] is generated. If the random number satisfy the following condition:

$$\partial \le e^{-\Delta E/T} \qquad (6.6)$$

then the newly produced solution is accepted as the current solution; else the current solution is unchanged.

Once a state (set of discrete variables) is selected based on the above criterion, the solution to the problem is obtained by using non-linear optimization.

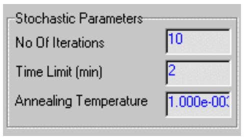
The Stochastic method consists of two groups:

- Stochastic Parameters
- Stochastic Optimization Output

Stochastic Parameters Group

The Stochastic Parameters consists of three parameters:

Figure 6.11

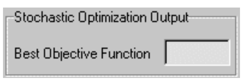


Parameter	Description
No Of Iterations	Allows you to specify the number of iteration in the simulated annealing algorithm. The number of iteration is a hard limit which means that the algorithm stops when the specified number of iteration is reached. By default, the No Of Iterations is set to 10.
Time Limit	Allows you to specify the algorithm time constraint (in minute). Once the Time Limit value is reached, the algorithm completes the move that it is solving. By default, the Time Limit is set at 2.
Annealing Temperature	Allows you to specify the temperature that is used to control the progression of the optimization problem toward an optimum solution. As a general rule, the Annealing Temperature should be the same order of magnitude of the Best Objective Function. By default, the Annealing Temperature is set to 1.000e-003.

Stochastic Optimization Output Group

In the Stochastic Optimization Output group, you can view the Best Objective Function value in the optimization based on the parameters specified in the Stochastic Parameters group.

Figure 6.12



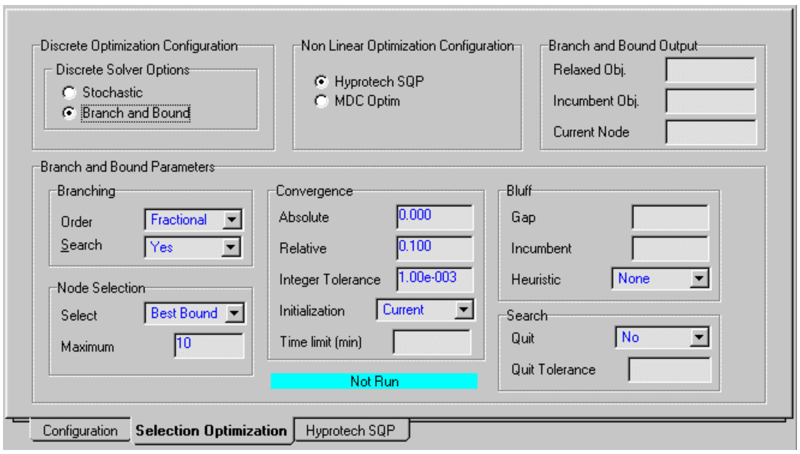
Branch and Bound Method

The Branch and Bound method first solves the original MINLP problem as a NLP problem by relaxing the integer restrictions, so that the calculations can converge with a less tight integer tolerance. The method continues by performing a systematic search of continuous solutions (called nodes), in which the integer variables are successively forced to take on integer values. This process is known as branching. The structure of this set of problems takes on the form of a tree. The procedure of branching, and solving a sequence of continuous problems is continued until a feasible integer solution is found. The value of the objective function becomes an upper bound of the objective of the MINLP problem. At this point, all of the continuous solutions whose objective function values are higher than the upper bound are eliminated from consideration. This elimination process is known as fathomed. Nodes are fathomed when the continuous problem is infeasible or when it has a natural integer solution. The search for the optimal solution terminates when all nodes are fathomed.

The Branch and Bound method contains two main groups:

- Branch and Bound Parameters
- Branch and Bound Output

Figure 6.13



Branch and Bound Parameters

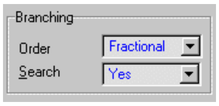
The Branch and Bound Parameters group consists of five sub groups:

- Branching
- Node Selection
- Convergence
- Bluff
- Search

Branching

The Branching group contains the following parameters:

Figure 6.14



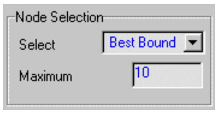
Parameters	Description
Order	Allows you to specify the method for selecting the branching variable. You can select the type of Order from the Order drop-down list: • **Fractional**. Selects the most fractional binary variable. • **Fixed**. Selects the variable by using the binary variable parameter rank. By default, the Order is set to Fractional.
Search	Allows you to indicate whether a branch-and-bound search is to be performed after the solution of the relaxed problem is found. By default, the Search is set to Yes.

Node Selection

A node is explicitly or implicitly fathomed when it satisfies one of the following conditions:

- when the solution is an integer value
- when the solution is infeasible
- when the optimal value is higher than the current upper bound

Figure 6.15



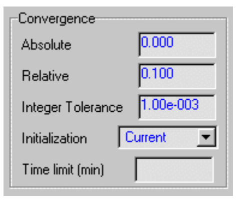
The Node Selection group contains the following parameters:

Parameters	Description
Select	Allows you to specify the method of node selection: • **Best Bound**. Selects the node with the lowest objective function value. • **Dive**. Selects the most recent node for branching. By default, Best Bound method is selected.
Maximum	Allows you to specify the maximum number of nodes to be searched (excluding relaxing and heuristic problems). The Maximum must be equal or larger than 1, and by default, it is set to 10.

Convergence

The Convergence group allows you to specify the integer restrictions and convergence conditions.

Figure 6.16



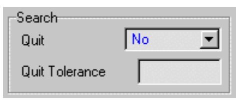
The Convergence group contains the following parameters:

Parameters	Description
Absolute	Allows you to specify the absolute convergence tolerance. The tolerance is compared with the absolute difference between the upper and lower bounds on the objective function. By default, the Absolute is set to 0.0.
Relative	Allows you to specify the relative convergence tolerance. The tolerance is compared with the relative difference between the upper and lower bounds on the objective function. The Relative value must be within the range of 0.0 to 1.0. By default, the Relative is set to 0.1.
Integer Tolerance	Allows you to specify the tolerance used when testing whether a relaxed binary variable is considered to be binary-valued (in other words, 0 or 1). The Integer Tolerance must be in between 0.0 to 0.5. By default, the Integer Tolerance is set to 1.0e-4.
Initialization	Allows you to specify the method of initializing optimization variables prior to the solution of each sub-problem. You can select one of following options from the Initialization drop-down list: • **Current**. Does not perform any re-initialization. In other words, variable values start with the values they had at the end of the previous sub-problem. • **Initial**. Sets variables to the values that they had when the algorithm was started. • **Relaxed**. Sets the variables to the optimal values found for the relaxed sub-problems. By default, the Initialization option is set to Current.
Time limit (min)	Allows you to specify a time limit (in minutes) for the search procedure. The Time limit is used in addition to the maximum number of nodes to place a limit on the length of the search. The Time limit value must be greater than 0.0. By default, the Time limit field is <empty>.

Search

The Search group contains the following parameters:

Figure 6.17

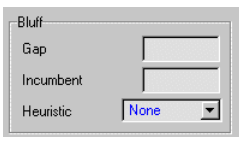


Parameters	Description
Quit	Allows you to end (Yes) or continue (No) the search when an improved integer solution over the Incumbent is found. By default, the Quit option is set to No.
Quit Tolerance	The Quit Tolerance provides the relative amount of improvement required in an integer solution to end the search. This parameter only has an effect if the Quit option is selected as Yes. The relative improvement in the Incumbent is defined as: $\frac{(F_I - f_I)}{\lvert F_I \rvert}$ *where:* F_I = *Objective function of the incumbent* f_I = *Objective function of the improved integer solution* If the Quit Tolerance field is specified as 0.0 or left empty, finding any improvement will end the search. By default, the Quit Tolerance is <empty>. You can only specify the Quit Tolerance value equal to or greater than 0.0.

Bluff

There are three parameters in the Bluff group:

Figure 6.18



- **Gap**. Provides an estimate of the relative gap between the objective function values of the relaxed problem, and optimal integer solution. The algorithm computes an upper bound on the objective function value after the solution of the relaxed problem. This is used to eliminate nodes in the search tree whose objective functions are not better than the Incumbent. The Incumbent objective function F_I is calculated from the fully relaxed objective F_R, and the Gap is defined as:

$$F_I = F_R + Gap \cdot (1.0 + |F_R|) \tag{6.7}$$

The branch-and-bound search will fail if the Gap value is too small. By default, the Gap field is <empty>. The Gap value must be greater than 0, and it is recommended to set it to 0.25.

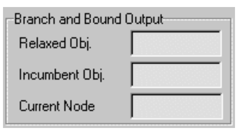
- **Incumbent**. The objective function value of the best binary-valued solution known so far. The value is used to eliminate nodes in the search that are not better than the Incumbent value. A value given for the Incumbent will override the value generated by using the Gap value. By default, the Incumbent field is <empty>.
- **Heuristic**. Allows you to specify the type of heuristic algorithm for finding an initial integer-feasible solution that is used to update the objective function Incumbent value. There are two types of Heuristic:
  - **Initial**. Uses initial State Variable values.
  - **Round**. Round the values of the State Variables.

By default, the Heuristic type is set to None. It is recommended that the type to be set to Initial.

Branch and Bound Output

The Branch and Bound Output group contains the following fields for displaying the optimization results:

Figure 6.19



Parameters	Description
Relaxed Obj.	Value of the objective function for the fully relaxed problem.
Incumbent Obj.	Value of the Incumbent objective function. The Incumbent is the lowest objective function of an integer-valued solution. This value is initialized by using the user-specified Incumbent and Gap parameters. The Incumbent Obj value is updated throughout the branch and bound search.
Current Node	The node of the branch and bound tree currently being solved. A number of 0 indicates that it is a fully relaxed problem.

6.4.2 Selection Optimization Tips

The following are setup tips for the Selection Optimization:

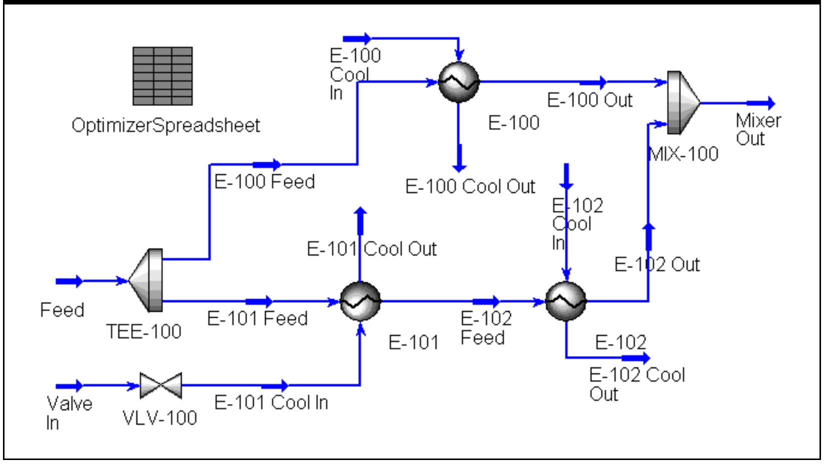
1. Since the algorithm used in the Stochastic method moves all over the solution space, the higher the number of iterations that the algorithm performs, the higher the probability there will be for finding a global optimum solution.
2. In the Branch and Bound method, reasonable convergence criteria, for example, value of Relative is important. It is recommended that you should accept the solution if an integer-feasible solution can be found during the search that has an objective function close to that of the fully relaxed solution.
3. In the Branch and Bound method, use Gap or Incumbent to reduce the size of the search space. Be aware that too much bluffing (for example a small value of Gap) can eliminate large sections of the search tree, resulting in a failed or sub-optimal search.
4. It is important to specify a Heuristic type in Branch and Bound method, especially if the Gap or Incumbent have not been specified. If the Heuristic successfully finds an integer-feasible solution, the objective function value can be used to eliminate sections of the search region. The Heuristic can satisfy the convergence criteria.

6.5 Example: Original Optimizer

Create the following sample case of multiple heat exchangers to optimize the overall UA by using the Original Optimizer.

PFD

Figure 6.20



Using the Peng Robinson property package and the listed components specify the process streams outlined in the following table.

Inlet Process Streams

Material Streams					
Tab [Page]	**In this cell...**	**Feed**	**E-100 Cool In**	**Valve In**	**E-102 Cool In**
Worksheet [Conditions]	Temperature (F)	20	-142	120	<empty>
	Pressure (psia)	1000	250	350	251
	Molar Flow (lbmole/hr)	2745	1542	<empty>	1640

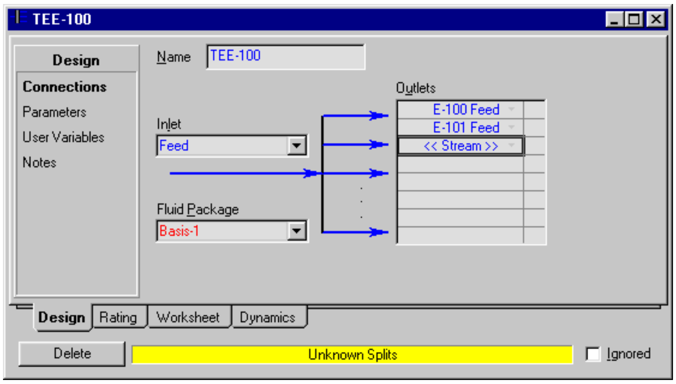
Material Streams					
Tab [Page]	**In this cell...**	**Feed**	**E-100 Cool In**	**Valve In**	**E-102 Cool In**
Worksheet [Composition]	Methane Mole Frac	0.7515	0.9073	0.0000	0.2828
	Ethane Mole Frac	0.2004	0.0927	0.0000	0.2930
	Propane Mole Frac	0.0401	0.0000	1.0000	0.1414
	i-Butane Mole Frac	0.0040	0.0000	0.0000	0.1313
	n-Butane Mole Frac	0.0040	0.0000	0.0000	0.1515

Process Operations

A tee, mixer, valve, and three heat exchangers are required for this process. Enter the data as shown in the figures below.

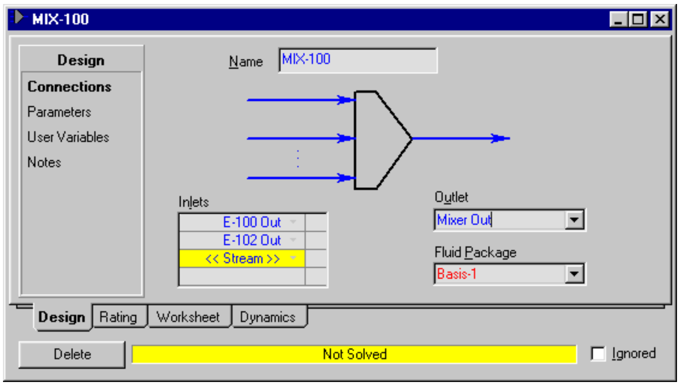
- TEE-100

Figure 6.21



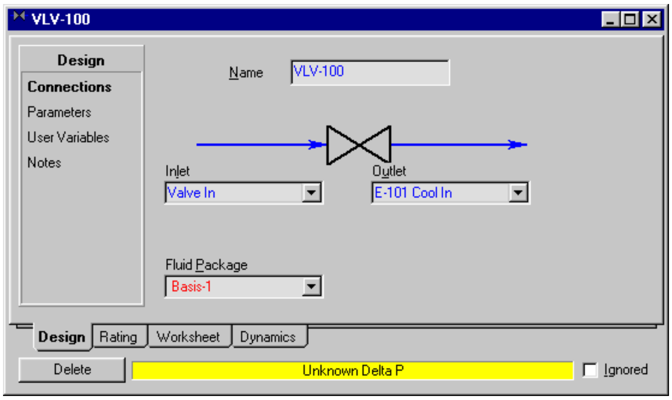
- MIX-100

Figure 6.22



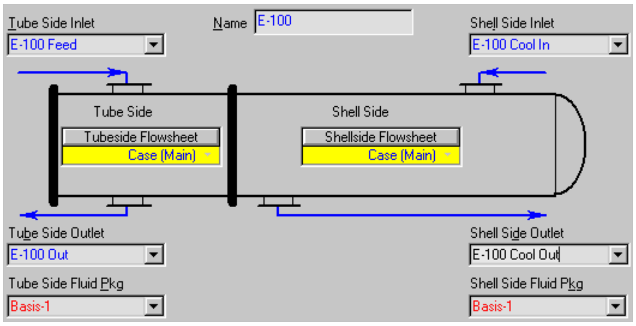
- VLV-100

Figure 6.23



• Heat Exchanger E-100

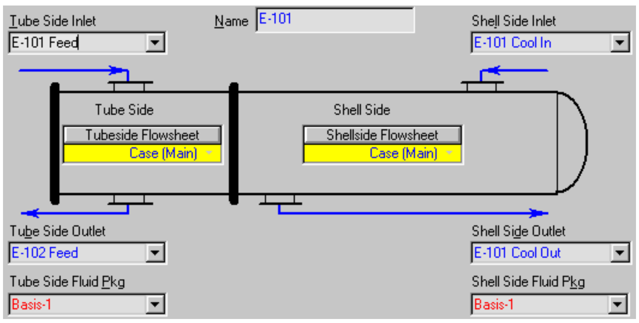
Figure 6.24



Heat Exchanger [E-100]		
Tab [Page]	**In this cell...**	**Enter**
Design [Parameters]	Tubeside Delta P	10 psi
	Shellside Delta P	10 psi
	UA	4.00e+04 Btu/F-hr
	Heat Leak/Loss	None
	Heat Exchange Model	Weighted
	Intervals (E-100 Feed)	10
	Intervals (E-100 Cool In)	10
	Dew/Bubble Pt (E-100 Cool In)	Inactive

• Heat Exchanger E-101

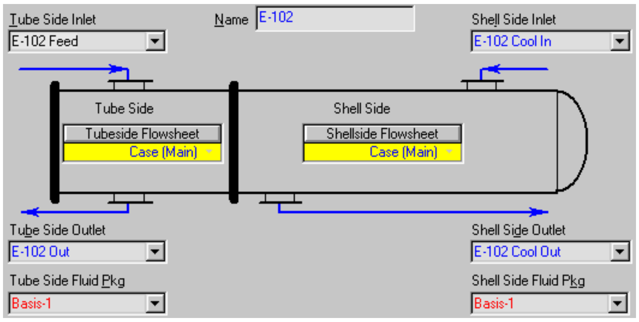
Figure 6.25



Heat Exchanger [E-101]		
Tab [Page]	**In this cell...**	**Enter**
Design [Parameters]	Tubeside Delta P	5 psi
	Shellside Delta P	1 psi
	UA	5.00e+04 Btu/F-hr
	Heat Leak/Loss	None
	Heat Exchange Model	Weighted
	Intervals (E-100 Feed)	10
	Intervals (E-100 Cool In)	10

- Heat Exchanger E-102

Figure 6.26



Heat Exchanger [E-102]		
Tab [Page]	**In this cell...**	**Enter**
Design [Parameters]	Tubeside Delta P	5 psi
	Shellside Delta P	5 psi
	UA	3.50e+04 Btu/F-hr
	Heat Leak/Loss	None
	Heat Exchange Model	Weighted
	Intervals (E-100 Feed)	10
	Intervals (E-100 Cool In)	10
	Dew/Bubble Pt (E-102 Cool In)	Inactive

Stream Specifications

- Temperature of stream E-102 Out, -40°F
- Vapour Fraction stream E-101 Cool Out, 1.00
- Temperature of stream E-100 Out, -65°F
- Pressure of E-101 Cool Out, 20 psia

Results

The calculated streams are shown in the figure below.

Figure 6.27

Name	Feed	E-100 Cool In	Valve In	E-102 Cool In	E-100 Feed
Vapour Fraction	1.0000	0.8249	0.0000	0.0363	1.0000
Temperature [F]	20.00	-142.0	120.0	-87.93	20.00
Pressure [psia]	1000	250.0	350.0	251.0	1000
Molar Flow [lbmole/hr]	2745	1542	375.4	1640	1077
Mass Flow [lb/hr]	5.577e+004	2.674e+004	1.655e+004	5.907e+004	2.187e+004
Liquid Volume Flow [barrel/day]	1.156e+004	5961	2237	9086	4535
Heat Flow [Btu/hr]	-9.752e+007	-5.471e+007	-1.887e+007	-8.315e+007	-3.825e+007
Name	E-101 Feed	E-100 Out	E-102 Out	Mixer Out	E-101 Cool In
Vapour Fraction	1.0000	0.0000	0.3714	0.0342	0.5234
Temperature [F]	20.00	-65.00	-40.00	-47.19	-28.68
Pressure [psia]	1000	990.0	990.0	990.0	21.00
Molar Flow [lbmole/hr]	1668	1077	1668	2745	375.4
Mass Flow [lb/hr]	3.389e+004	2.187e+004	3.389e+004	5.577e+004	1.655e+004
Liquid Volume Flow [barrel/day]	7027	4535	7027	1.156e+004	2237
Heat Flow [Btu/hr]	-5.927e+007	-4.105e+007	-6.224e+007	-1.033e+008	-1.887e+007
Name	E-100 Cool Out	E-102 Feed	E-101 Cool Out	E-102 Cool Out	** New **
Vapour Fraction	1.0000	0.8456	1.0000	0.1583	
Temperature [F]	-21.68	-13.30	-30.84	-56.17	
Pressure [psia]	240.0	995.0	20.00	246.0	
Molar Flow [lbmole/hr]	1542	1668	375.4	1640	
Mass Flow [lb/hr]	2.674e+004	3.389e+004	1.655e+004	5.907e+004	
Liquid Volume Flow [barrel/day]	5961	7027	2237	9086	
Heat Flow [Btu/hr]	-5.191e+007	-6.067e+007	-1.747e+007	-8.158e+007	

6.5.1 Optimizing Overall UA

The Optimizer determines the optimum Tee flow ratio such that the Overall UA is minimized. Therefore, delete the individual heat exchanger UA specs and replace them with the following:

- Temperature of E-102 Cool In = -85°F
- Flowrate of Valve In = 495 lbmole/hr
- Flowrate of E-101 Feed = Optimized variable (Initially set to the previous flow rate of 1,670 lbmole/hr)

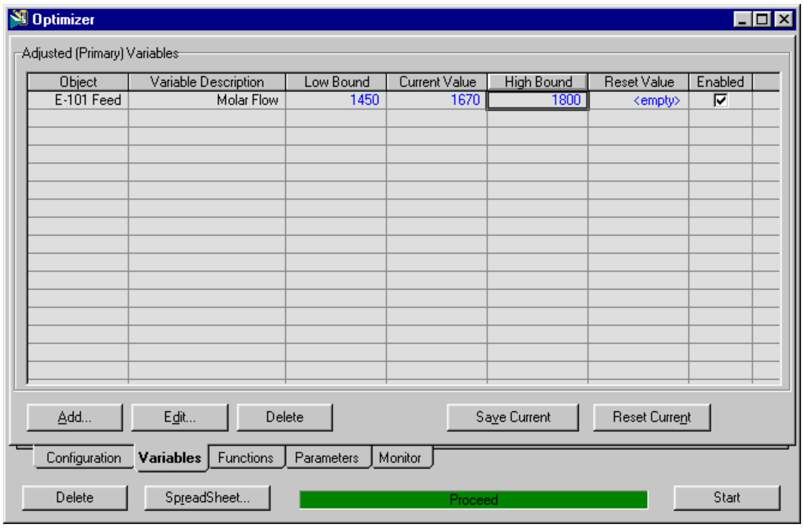
After replacing the specs, the flowsheet solves and UAs are calculated.

Opening the Optimizer

To open the Optimizer:

1. Click the **Add** button in the **Variables** tab to display the Variable Navigator.
2. Select the Molar Flow of the stream E-101 Feed.
3. Specify the Low and High Bounds as shown in the figure below.

Figure 6.28



The search is now within a range of 1450 lbmole/hr - 1800 lbmole/hr to avoid a temperature cross.

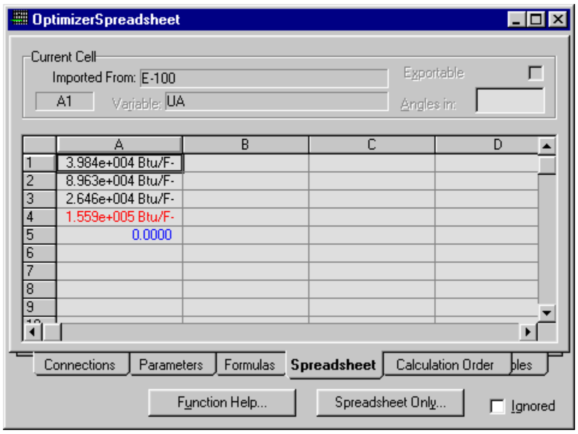
Import the Heat Exchanger

To import the three Heat Exchanger UAs to the Optimizer spreadsheet:

1. Click the **SpreadSheet** button to display the Optimizer Spreadsheet property view.
2. Click the **Connections** tab and then click the **Add Import** button to display the Variable Navigator property view.
3. Import the UA value for the heat exchanger E-100.

4. Repeat steps#2 and#3 for the Heat Exchangers E-101 and E-102.
5. Click the **Spreadsheet** tab. In cell A4, enter the formula, **+a1+a2+a3**. This sums the UAs. In cell A5, enter 0.0. This is used in the constraints.

Figure 6.29



6. Close the Optimizer Spreadsheet property view.

Defining the Objective Function

You must define the Objective Function and the Constraint Functions. The Objective Function is the expression being minimized, which in this case is the sum of the Heat Exchanger UAs.

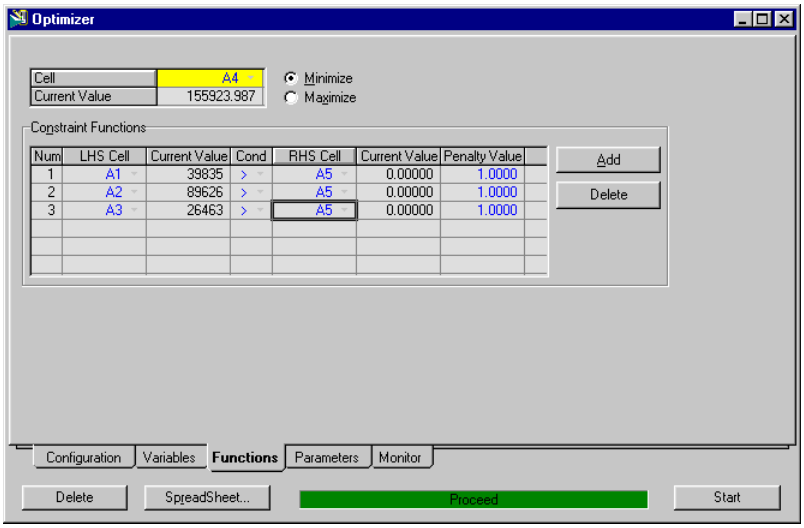
1. Click the **Functions** tab in the Optimizer property view.
2. Click the **Cell** drop-down list and select **A4**. The value of the cell is displayed in the **Current Value** field.
3. Click the **Minimize** radio button.

Adding Constraint Functions

Enter constraint functions to ensure the solution is reasonable. Each Heat Exchanger UA must be greater than zero.

1. Click the **Add** button three times to add three constraints to the table.
2. In the LHS Cell drop-down list, select the cell A1, A2, and A3 for each of the respective constraints.
3. In the RHS Cell drop-down list, select the cell A5 for each of the constraints.

Figure 6.30



4. Click the **Parameters** tab. For this example, use the **Mixed** method leaving all the parameters at their defaults.
5. Click the **Start** button, and then click the **Monitor** tab to watch the progress of the Optimizer.

An optimum molar flow of 1,800 lbmole/hr is obtained for the stream E-101 Feed, corresponding to an overall UA of about 1.43e5 Btu/F-hr. This compares to the specified value of 1.5e5 Btu/F-hr in the first part of this example.

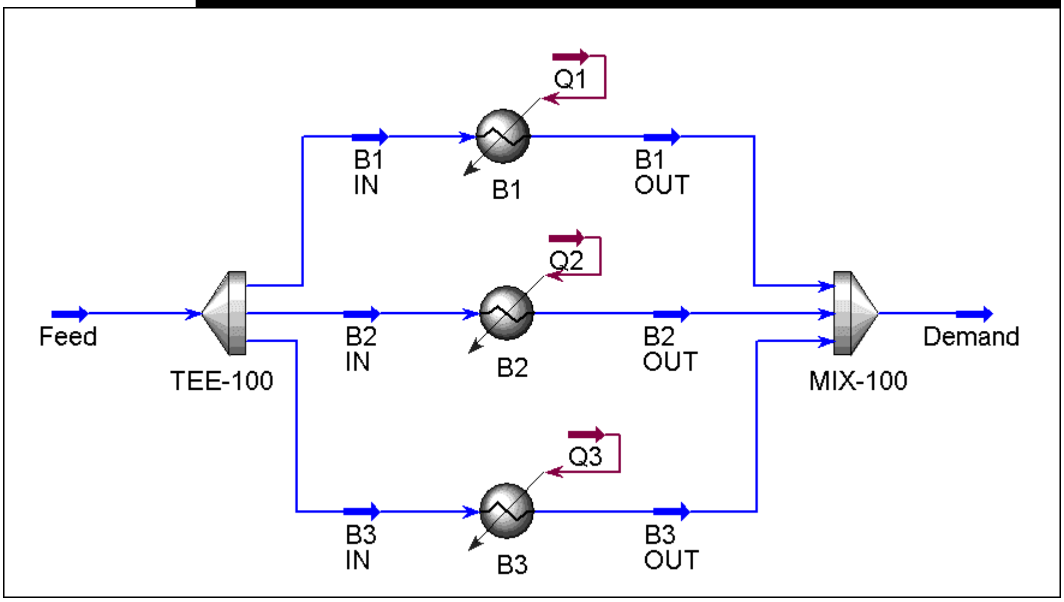
6.6 Example: MNLP Optimization

In this example, the Hyprotech SQP Optimizer is used with Selection Optimization to determine the most economical use of each boiler in a steam utility system to meet the steam demand.

Create the following steam utility system in Steady-State.

PFD

Figure 6.31



The steam demand is supplied by using three parallel boilers connected to a common high-pressure header. The boilers are modeled as simple heaters, each with different capacity, fuel cost, overhead cost, and efficiency. The high-pressure header is modeled by using a tee unit operation.

Defining the Simulation Basis

Refer to **Section 5.2 - Simulation Basis Manager** in the **HYSYS User Guide** for more information on selecting a property package, and adding components.

1. In the Simulation Basis Manager, define the property package as NBS Steam, and specify water (H2O) as the component.
2. Specify the Feed stream as follows:

In this cell...	Enter...
Temperature	21.3°C (70.3°F)
Pressure	4,101.3 kPa (594.8 psia)
Mass Flow	14,400 kg/h (31746 lb/hr)
Mass Fraction (H2O)	1

3. Specify the mass flowrate for the following boiler inlet streams:

Stream	Mass Flowrate
B2 IN	3,600 kg/h (7937 lb/hr)
B3 IN	5,400 kg/h (11905 lb/hr)

The mass flowrate of B1 IN is 5,400 kg/h, which is automatically calculated by HYSYS after you have defined B2 IN and B3 IN.

4. Specify a temperature of 350°C (662°F) for all three boilers outlet streams (B1 OUT, B2 OUT, and B3 OUT).
5. Specify a pressure drop of 0 kPa for each boiler.

The steam utility system is now fully defined. The results of each stream are summarized in the Workbook as shown below:

Figure 6.32

Name	Feed	B1 IN	B2 IN	B3 IN
Vapour Fraction	0.0000	0.0000	0.0000	0.0000
Temperature [C]	21.28	21.28	21.28	21.28
Pressure [kPa]	4101	4101	4101	4101
Molar Flow [kgmole/h]	799.3	299.7	199.8	299.7
Mass Flow [kg/h]	1.440e+004	5400	3600	5400
Liquid Volume Flow [m3/h]	14.43	5.411	3.607	5.411
Heat Flow [kJ/h]	-2.280e+008	-8.549e+007	-5.699e+007	-8.549e+007

Name	B1 OUT	B2 OUT	B3 OUT	Demand
Vapour Fraction	1.0000	1.0000	1.0000	1.0000
Temperature [C]	350.0	350.0	350.0	350.0
Pressure [kPa]	4101	4101	4101	4101
Molar Flow [kgmole/h]	299.7	199.8	299.7	799.3
Mass Flow [kg/h]	5400	3600	5400	1.440e+004
Liquid Volume Flow [m3/h]	5.411	3.607	5.411	14.43
Heat Flow [kJ/h]	-6.931e+007	-4.620e+007	-6.931e+007	-1.848e+008

Heat Flow & Costs Calculations

Refer to **Section 5.10 - Spreadsheet** for more information on the Spreadsheet operation.

Before the case is converted into an optimization problem, the efficiency, actual heat flow, and operating cost for each boiler are calculated within the Spreadsheet operation.

Add a Spreadsheet operation to the case and name it to Boilers Calculations.

Efficiency

The efficiency of the three boilers are locally characterized by the following relationship:

$$Eff = Eff_{MAX} - 5\% \times \left(\frac{Flow - Flow_{MAX}}{Flow_{MAX-5\%} - Flow_{MAX}}\right)^2 \qquad (6.8)$$

where:

Eff = Efficiency of the boiler

Eff_{MAX} = Maximum efficiency of the boiler

The quadratic approximation between the steam mass flowrate, and boiler efficiency is valid only within a narrow range of operation, localized near the point of maximum efficiency.

$Flow$ = Inlet mass flowrate of the boiler

$Flow_{MAX}$ = Inlet mass flowrate of the boiler at maximum efficiency

$Flow_{MAX-5\%}$ = Inlet mass flowrate at an efficiency 5% less than the maximum efficiency.

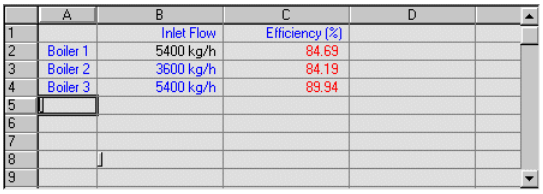
The squared term in **Equation (6.8)** is a scaled deviation in flowrate with respect to the mass flow which gives the maximum efficiency.

The following table lists the efficiency parameters associated with each boiler:

Parameters	B1	B2	B3
Eff_{MAX}	85%	87%	90%
$Flow_{MAX}$	1.8 kg/s	2.2 kg/2	1.6 kg/s
$Flow_{MAX-5\%}$	3.0 kg/s	3.8 kg/s	2.5 kg/s

Calculate the efficiency of each boiler in the Boilers Calculations spreadsheet. The following efficiency results should appear on the Spreadsheet:

Figure 6.33



	A	B	C	D
1		Inlet Flow	Efficiency (%)	
2	Boiler 1	5400 kg/h	84.69	
3	Boiler 2	3600 kg/h	84.19	
4	Boiler 3	5400 kg/h	89.94	
5				
6				
7				
8				
9				

Actual Heat Flow

The heat flow values calculated in HYSYS are the heat flow required by each boiler when the boiler is operating at the efficiency calculated from **Equation (6.8)**. Therefore, the actual heat flow is calculated with the following equation:

$$\text{Actual Heat Flow} = \text{Heat Flow} \times \frac{100}{Eff} \tag{6.9}$$

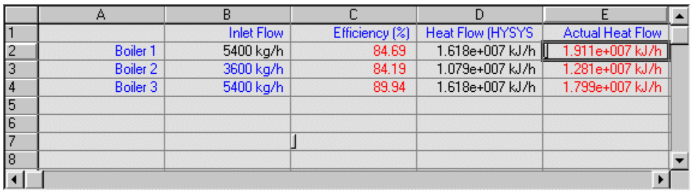
where:

Heat Flow = Heat flow of the boiler calculated in HYSYS

Eff = Efficiency of the boiler (see **Equation (6.8)**)

The calculated efficiencies are used to calculate the Actual Heat Flow required by each boiler. The following results should appear on the Boilers Calculations spreadsheet:

Figure 6.34



	A	B	C	D	E
1		Inlet Flow	Efficiency (%)	Heat Flow (HYSYS	Actual Heat Flow
2	Boiler 1	5400 kg/h	84.69	1.618e+007 kJ/h	1.911e+007 kJ/h
3	Boiler 2	3600 kg/h	84.19	1.079e+007 kJ/h	1.281e+007 kJ/h
4	Boiler 3	5400 kg/h	89.94	1.618e+007 kJ/h	1.799e+007 kJ/h
5					
6					
7					
8					

Cost Calculations

The objective of this optimization problem is to minimize the total operating cost of the system, which is defined as the additive operating costs of each boiler:

$$\text{Total Operating Cost} = \sum(\text{Operating Cost})_i \qquad (6.10)$$

where:

i = Boiler i

The operating cost for each boiler is the sum of the fuel consumption cost, and overhead cost associated with the operation. The fuel cost, and the overhead cost of each boiler are listed in the following table:

Cost	B1	B2	B3
Fuel ($/MJ)	0.008	0.0085	0.0088
Overhead ($/hr)	30	29	25

Since the overhead cost only applies when the boiler is operating, the overhead cost is multiplied to a binary state variable of 1 or 0; a value of 1 indicates an ‘on’ status, and 0 reflects a ‘off’ status. The operating cost for each boiler is defined as follows:

$$\text{Operating Cost} = \begin{pmatrix}\text{Actual Heat}\\ \text{Flow}\end{pmatrix} \times \begin{pmatrix}\text{Fuel}\\ \text{Cost}\end{pmatrix} + \begin{pmatrix}\text{Overhead}\\ \text{Cost}\end{pmatrix} \times \text{Status} \qquad (6.11)$$

where:

Status = Binary state variable of the boiler (1 = on, 0 = off)

To specify an ‘on’ status for all three boilers in the Boilers Calculations spreadsheet:

1. Create a new column called **Status**.
2. Specify a value of **1** in three spreadsheet cells under the Status column to represent the state of the three boilers.

For now, it is assumed that all three boilers are in operation. Therefore the overhead cost applies to all three boilers. During optimization, the Status of each boiler changes accordingly to obtain the minimum objective function.

The following operating cost for each boiler should appear on the Boilers Calculations spreadsheet:

Figure 6.35

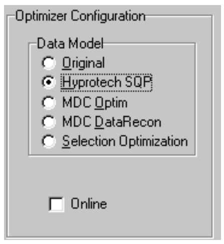
	A	B	C	D	E
5					
6		Status	Fuel Cost ($/MJ)	Overhead Cost ($/h)	Operating Cost ($/h)
7	Boiler 1	1.000	8.000e-003	30.00	182.9
8	Boiler 2	1.000	8.500e-003	29.00	137.9
9	Boiler 3	1.000	8.800e-003	25.00	183.3
10					
11				Total Operating Cost	504.1
12					

6.6.1 NLP Setup

Defining the Optimizer

1. Select **Optimizer** from the **Simulation** menu. The Optimizer property view appears.
2. In the Optimizer property view, select the **Hyprotech SQP** radio button on the **Configuration** tab.

Figure 6.36



Refer to **Section 6.3 - Hyprotech SQP Optimizer** for more information on Accuracy Tolerance and Perturbation.

3. Click on the **Hyprotech SQP** tab.
4. In the Configuration group, click on the **Setup** radio button.
5. In the Setup group, set the Accuracy Tolerance to **1.00e-006**, and set the Perturbation to **1.00e-004**.
6. Close the Optimizer property view.

Adding the Derivative Utility

1. Select **Utilities** from the **Tools** menu.
2. Select **Derivative Utilities**.

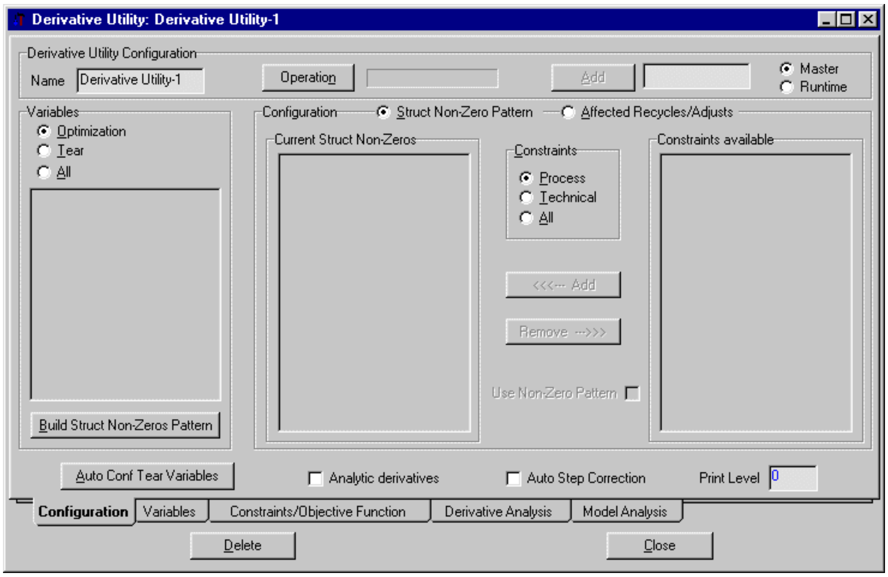
Refer to **Section 7.26 - Utilities** in the **HYSYS User Guide** for more information on Utilities.

3. Click the **Add Utility** button to add a Derivative Utility. The Derivative Utility property view appears.

Defining the Object Filter

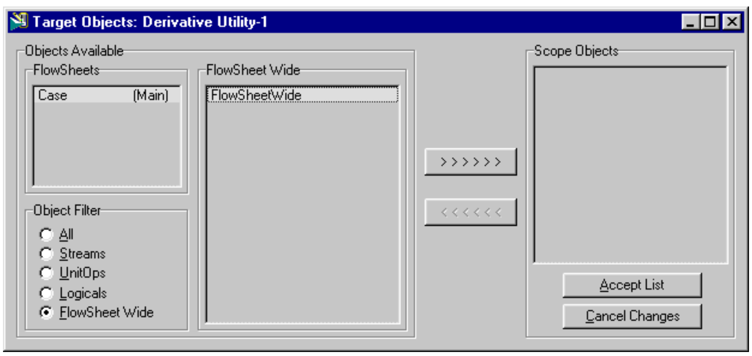
1. Click on the **Configuration** tab in the Derivative Utility property view.

Figure 6.37

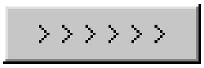


2. In the Derivative Utility Configuration group, click on the **Operation** button. The Target Objects property view appears.

Figure 6.38



3. Click on the **Flowsheet Wide** radio button in the Object Filter Group.
4. Select **FlowsheetWide** in the Flowsheet Wide group.

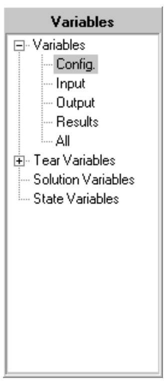


Transfer icon

5. Click the **Transfer** icon to transfer **FlowsheetWide** to the Scope Objects group.
6. Click the **Accept List** button in the Scope Objects group to save the setting. The Target Objects property view closes.

Defining the Optimization Variables

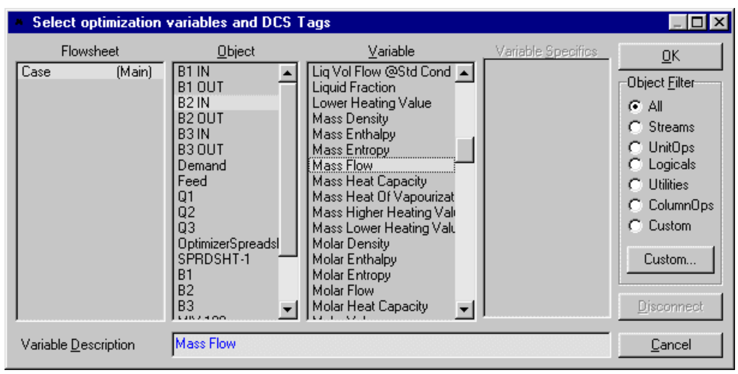
1. In the Derivative Utility property view, click on the **Variables** tab.
2. In the Variables tree browser, select the **Variables** branch.
3. Click on the **Plus** icon ⊞ to expand the Variables branch.



Variables tree browser

4. Select the **Config.** sub branch.
5. In the Derivative Utility Configuration group, select **OptVars** from the Add drop-down list.
6. Click on the **Add** button. The Select optimization variables and DCS Tags property view appears.
7. Select **B2 IN** from the Object list.
8. Select **Mass Flow** from the Variable list.

Figure 6.39



9. Click **OK**.

   The mass flowrate of B2 IN is added to the Variables Config table on the Variables tab in the Derivative Utility property view.

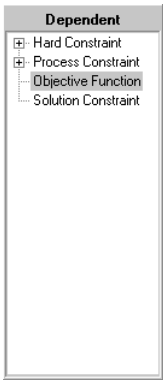
10. Rename the Object Name to **B2 IN** as shown below:

Figure 6.40

	Object Name	Attached Object	Property	Current Value	Optimize Flag
B2 IN	B2 IN	B2 IN	MassFlow	3600.0000	☑

11. Click on the **Input** sub branch under the Variables branch in the Variables group.
12. In the Variables Input table, set the Minimum, and Maximum of B2 IN to **0**, and **36,000 kg/h**, respectively.
13. Repeat step 2 to 12 for **B3 IN**.

Defining the Objective Function



Dependent tree browser

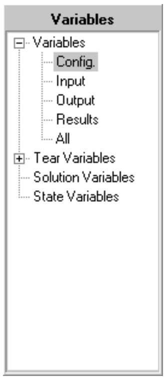
1. In the Derivative Utility property view, click on the **Constraints/Objective Functions** tab.
2. In the Dependent tree browser, select the Objective Function branch.
3. In the Derivative Utility Configuration group, select **ObjFunc** from the Add drop-down list.
4. Click the **Add** button. The Select optimization variables and DCS Tags property view appears.
5. Select **Boilers Calculations** as the Object.
6. From the Variable list, select the cell where you calculated the total operating costs of all three boilers.
7. Click **OK**.

   The Total Operating Cost value appears in the Objective Function table on the Constraints/Objective Function tab.
8. Set the Price to 1.
9. Change the Object Name to Total Cost as shown below.

Figure 6.41

	Object Name	Attached Object	Property	Current Value	Current	Price
Total Cost	Total Cost	Boilers Calculati	ExtraData	470.1808	470.1808	1.0000

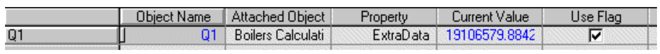
Defining the Process Constraints



Dependent tree browser

1. In the Derivative Utility property view, click on the **Constraints/Objective Function** tab.
2. Expand the Process Constraints branch from the Dependent tree browser.
3. Select the **Config.** sub branch.
4. In the Derivative Utility Configuration group, select **ProcCons** from the Add drop-down list.
5. Click the **Add** button.
6. Select the **Boilers Calculations** spreadsheet from the Variable list.
7. From the Object List, select the cell where you calculated the actual heat flow for boiler 1.
8. Click **OK**.
9. In the Process Constraint Config table, change the Object Name of the newly added process constraint to **Q1** as shown below:

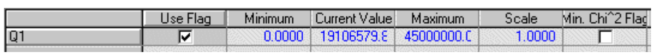
Figure 6.42



	Object Name	Attached Object	Property	Current Value	Use Flag
Q1	Q1	Boilers Calculati	ExtraData	19106579.8842	☑

10. In the Dependent group, select the **Input** sub branch under the Process Constraints branch.
11. Set the Scale to 1.
12. Specify a Minimum of 0 and Maximum of 12,500 kW as show below:

Figure 6.43



	Use Flag	Minimum	Current Value	Maximum	Scale	Min. Chi^2 Flag
Q1	☑	0.0000	19106579.8	45000000.0	1.0000	☐

13. Select the **Use Flag** checkbox.

    The Use Flag checkbox allows you to eliminate evaluation of infeasible integer candidate. You can reduce the amount of time for the optimization calculation by selecting the appropriate variables.

14. Add a series of process constraints as shown in the table below by repeating step 2 to 13 for each constraint.

Variable	Object	Object Name	Minimum	Maximum
B2	Heat Flow	Q2	0 kW	9,500 kW
B3	Heat Flow	Q3	0 kW	13,500 kW
B1 IN	Mass Flow	B1 IN	0 kg/s	10 kg/s
B2 IN	Mass Flow	B2 IN	0 kg/s	10 kg/s
B3 IN	Mass Flow	B3 IN	0 kg/s	10 kg/s

6.6.2 MINLP Setup

Defining Slack Variables

Three slack variables are added to the Boilers Calculations spreadsheet to represent the maximum flow constraints on the inlet mass flowrate of each boiler. The slack variable is defined as follows:

$$Slack_{MAX} = Flow - Flow_{MAX} \times Status \tag{6.12}$$

where:

$Slack_{MAX}$ = Maximum slack value

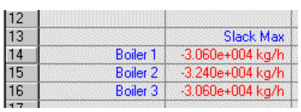
Flow = Inlet mass flowrate of the boiler

$Flow_{MAX}$ = Maximum inlet mass flowrate of the boiler (36,000 kg/h)

1. Calculate the slack values for the three boilers.

   The following slack values should appear in the Boilers Calculations spreadsheet:

Figure 6.44



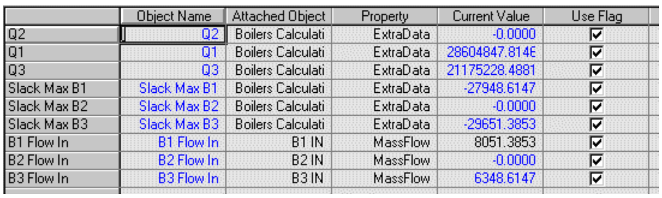
12		
13		Slack Max
14	Boiler 1	-3.060e+004 kg/h
15	Boiler 2	-3.240e+004 kg/h
16	Boiler 3	-3.060e+004 kg/h

Refer to the section on **Defining the Process Constraints** for more information on adding process constraints in the Derivative Utility.

2. Add the slack values from the Boilers Calculations spreadsheet to the Derivative Utility as process constraints.
3. In the Derivative Utility, set the Maximum of each slack variable to 0.
4. Set the Minimum to a superfluous value of -1.00e+006.
5. Rename the slack values as Slack Max B1, Slack Max B2, and Slack Max B3 for the appropriate boiler
6. Select the **Use Flag** checkbox for all slack variables.

The Process Constraints table should contain all the constraints as shown in the following figure:

Figure 6.45



	Object Name	Attached Object	Property	Current Value	Use Flag
Q2	Q2	Boilers Calculati	ExtraData	-0.0000	☑
Q1	Q1	Boilers Calculati	ExtraData	28604847.814E	☑
Q3	Q3	Boilers Calculati	ExtraData	21175228.4881	☑
Slack Max B1	Slack Max B1	Boilers Calculati	ExtraData	-27948.6147	☑
Slack Max B2	Slack Max B2	Boilers Calculati	ExtraData	-0.0000	☑
Slack Max B3	Slack Max B3	Boilers Calculati	ExtraData	-29651.3853	☑
B1 Flow In	B1 Flow In	B1 IN	MassFlow	8051.3853	☑
B2 Flow In	B2 Flow In	B2 IN	MassFlow	-0.0000	☑
B3 Flow In	B3 Flow In	B3 IN	MassFlow	6348.6147	☑

Defining the State Variables

Define three state variables in the Derivative Utility to reference the three binary equipment states from the Boilers Calculations spreadsheet.

To add the state variables in the Derivative Utility:

1. In the Derivative Utility property view, click on the **Variables** tab.
2. In the Variables group, select the State Variable branch.
3. Select **StateVars** from the Add drop-down list in the Derivative Utility Configuration group.
4. Click **Add**.
5. Select **Boilers Calculations** as the Object.
6. From the Object List, select the cell where you specified the binary state variable for Boiler 1.
7. Click **OK**.
8. Rename the newly added state variable to Use B1.

9. Repeat step 3 to 8 for Boiler 2, and Boiler 3.

   The State Variables table should appear as follows:

Figure 6.46

	Object Name	Attached Object	Status	Rank
Use B1	Use B1	Boilers Calculations@	On	0
Use B2	Use B2	Boilers Calculations@	On	0
Use B3	Use B3	Boilers Calculations@	On	0

10. Click the **Close** button to close the Derivative Utility property view.

Defining the Selection Optimization

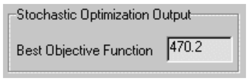
The final step is to define the Selection Optimization requirements.

1. Select **Optimizer** from the **Simulation** menu.
2. Click on the **Selection Optimization** radio button.
3. Click on the **Selection Optimization** tab.
4. Click on the **Stochastic** radio button in the Discrete Solver Options group.
5. In the Stochastic Parameters group, set the Time Limit (min) to **5**.
6. Set the Annealing Temperature to **100**.
7. Click the **Start** button.

Optimization Results

The lowest cost (best objective function value) is displayed in the Stochastic Optimization Output Group on the Selection Optimization tab of the Optimizer property view:

Figure 6.47



The flow, and operating conditions of each boiler required to achieve the best objective function value are summarized in the Boilers Calculations spreadsheet as shown below:

Figure 6.48

	A	B	C	D	E
1		Inlet Flow	Efficiency (%)	Heat Flow	Actual Heat Flow
2	Boiler 1	8051 kg/h	84.34	2.412e+007 kJ/h	2.860e+007 kJ/h
3	Boiler 2	-9.992e-014 kg/h	77.55	-2.994e-010 kJ/h	-3.861e-010 kJ/h
4	Boiler 3	6349 kg/h	89.83	1.902e+007 kJ/h	2.118e+007 kJ/h
5					
6		Status	Fuel Cost	Overhead Cost	Operating Cost
7	Boiler 1	1.000	8.000e-003	30.00	258.8 kJ/h
8	Boiler 2	0.0000	8.500e-003	29.00	-3.282e-015 kJ/h
9	Boiler 3	1.000	8.800e-003	25.00	211.3 kJ/h
10					
11				Total Operating Cost	470.2 kJ/h
12					
13		Slack Max			
14	Boiler 1	-2.795e+004 kg/h			
15	Boiler 2	-9.992e-014 kg/h			
16	Boiler 3	-2.965e+004 kg/h			

6.7 References

[1] Box, M.J. "A New method of Constrained Optimization and a Comparison with other Methods," Computer J., 8, 42-45, 1965.

[2] Press, W.H., et al, "Numerical Recipes in C," Cambridge university Press, 1988.

[3] Kuester, J.L. and Mize, J.H., "Optimization Techniques with FORTRAN," McGraw-Hill Book Co., 1973.

[4] Harwell Subroutine Library, Release 10, Advanced Computing Dept., AEA Industrial Technology, Harwell laboratory, England, 1990.

[5] Powell, M.J.D., "A Fast Algorithm for Non-Linearly Constrained Optimization Calculations," Numerical Analysis, Dundee, 1977, Lecture Notes in Math. 630, Springer-Verlag, 1978.

[6] Chamberlain R.M. and Powell, M.J.D., "The Watchdog Technique for Forcing Convergence in Algorithms for Constrained Optimization," Mathematical Programming Study, 16, 1-17, 1982.

7 Piping Operations

7.1 Compressible Gas Pipe

The Compressible Gas Pipe (CGP) model uses an algorithm that solves a vector system using the Two-Step Lax-Wendroff method with Boris & Book anti-diffusion.

The CGP unit operation is primarily designed for transient calculations with streams. Steady state calculations have been implemented primarily for initialization of the Pipe State prior to transient calculations.

The following calculation modes are supported in steady sate mode:

- Specify Inlet Pressure, Temperature, and Mass Flow
- Specify Inlet Temperature, Mass Flow, and Outlet Pressure
- Specify Inlet Pressure and Temperature, and Outlet Pressure. Alternatively the pressure drop may be used with either boundary pressure.

Model for a Single Phase Compressible Flow

The following equations are used in HYSYS to model a single phase compressible flow.

Governing Equations

- Mass:

$$\frac{\partial(A\rho)}{\partial t} + \frac{\partial(A\rho u)}{\partial x} = 0 \tag{7.1}$$

- Momentum:

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2 + p)}{\partial x} = \rho g \sin\theta - \frac{1}{2} f\rho u|u|\frac{S}{A} - \rho u^2 \frac{1}{A}\frac{dA}{dx} \tag{7.2}$$

- Energy:

$$\frac{\partial(\rho E)}{\partial t} + \frac{\partial(\rho Hu)}{\partial x} = k(T_{wall} - T)\frac{S}{A} - \rho g \sin\theta - \frac{1}{2}f\rho u^2|u|\frac{S}{A} - \rho Hu\frac{1}{A}\frac{dA}{dx} \tag{7.3}$$

where:

$A = \frac{1}{4}\pi D^2$, *pipe cross-sectional area*

$E = e + \frac{1}{2}u^2$, *total internal energy*

$H = h + \frac{1}{2}u^2$, *total enthalpy*

$S = \pi D$, *the pipe perimeter*

D = *pipe diameter*

e = *internal energy*

f = *friction factor*

g = *acceleration due to gravity*

h = *enthalpy*

k = *heat transfer coefficient*

p = *pressure*

t = *time*

T = *temperature*

T_{wall} = *wall temperature*

u = *velocity*

x = *distance*

θ = *pipe inclination*

ρ = *density*

Algorithm

The algorithm solves the vector system by the Two-Step Lax-Wendroff method with Boris & Book anti-diffusion.

$$\frac{\partial U}{\partial t} + \frac{\partial D}{\partial x} = \underline{G} \qquad (7.4)$$

7.1.1 Compressible Gas Pipe Property View

There are two ways that you can add a Compressible Gas Pipe to your simulation:

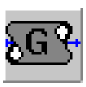
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Piping Equipment** radio button.
3. From the list of available unit operations, select **Compressible Gas Pipe**.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

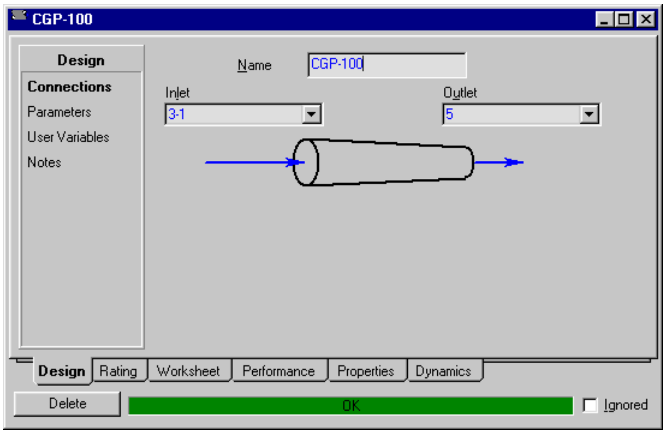
   You can also open the Object Palette by pressing **F4**.
2. Double-click the **Compressible Gas Pipe** icon.



Compressible Gas Pipe icon

The Compressible Gas Pipe property view appears.

Figure 7.1



7.1.2 Design Tab

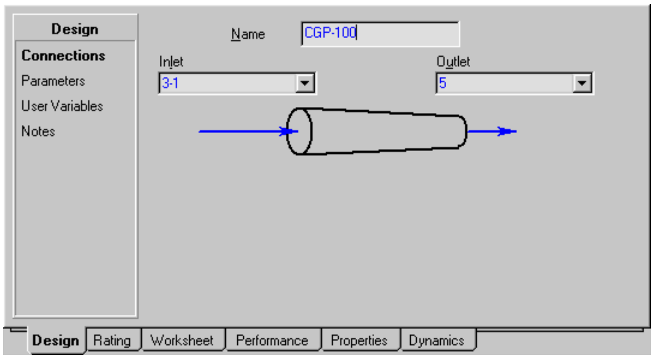
The Design tab contains the following pages:

- Connections
- Parameters
- User Variables
- Notes

Connections Page

On the Connections page, you must specify the Feed and Product material streams.

Figure 7.2



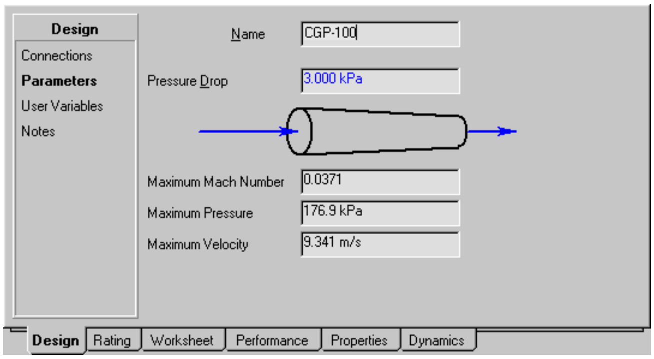
You can specify the streams by either typing the name of the new stream or selecting existing streams in the Inlet and Outlet drop-down lists. You can also edit the name of the operation on this page.

The Compressible Gas Pipe does not support an energy stream.

Parameters Page

The Parameters page allows you to specify the pressure drop across the pipe as well as the name of the operation.

Figure 7.3



There are also three calculated values that are displayed on the page.

- **Max. Mach Number**. For steady state calculations this is always at the outflow from the pipe. During dynamic calculations this can be at any location within the pipe.
- **Max. Pressure**. For steady state calculations this is always at the outflow from the pipe. During dynamic calculations this can be at any location within the pipe.
- **Max. Velocity**. For steady state calculations this is always at the outflow from the pipe. During dynamic calculations this can be at any location within the pipe.

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

7.1.3 Rating Tab

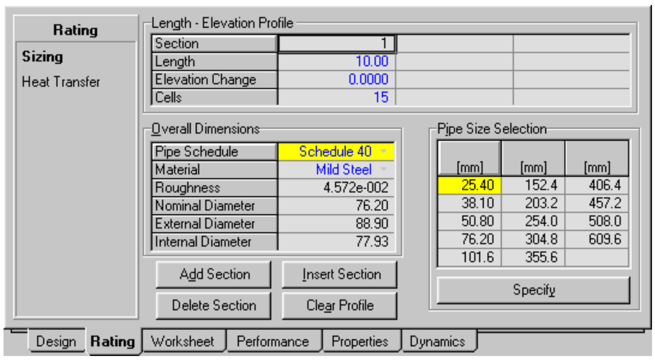
The Rating tab consists of two pages:

- **Sizing**. you provide information regarding the dimensions of sections in the pipe segment
- **Heat Transfer**. the heat loss of the pipe segment can either be specified or calculated from various heat transfer parameters.

Sizing Page

On the Sizing page, the length-elevation profile for the CGP is constructed. You can provide details for each fitting or pipe section that is contained in the CGP that you are modeling. An unlimited number of pipe sections or fittings can be added on this page.

Figure 7.4



For a given length of pipe which is modelled in HYSYS, the parameters of each section is entered separately. To fully define the pipe section, you must also specify pipe schedule, diameters (nominal or inner and outer), a material, and a number of cells.

There are two ways that you can add sections to the length-elevation profile:

- Click the **Add Section** button, which allows you to add the new section after the currently selected section.
- Click the **Insert Section** button, which allows you to add the new section before the currently selected section

For each segment that you add, you must specify the following:

- **Length**. The physical length of the pipe. Notice that it is not appropriate to enter an equivalent length and attempt to model fittings.
- **Elevation Change**. The elevation change of the pipe.
- **Cells**. Number of cells within the pipe (10 - 1000).

When modeling multiple sections, faster and more stable convergence can be obtained if all cell sizes are similar. For a stable solution, the number of cells should be selected such that the following constraint is met:

$$\frac{\text{Cell Length}}{\text{Time Step}} < 0.5\ \text{Sonic Velocity} \tag{7.5}$$

The cells have to be sufficiently small to ensure that in any one time step there will be changes of sufficient magnitude in a sufficient number of cells to ensure that the solver used by the Compressible Gas pipe and the HYSYS dynamics pressure-flow solver interact correctly.

To delete a section, click on the section you want to delete and click the Delete button. The Clear Profile button deletes all sections except for the first section, however, all data for the first section is cleared.

Refer to **Section 7.3 - Pipe Segment** for more information.

The Overall Dimensions group manages the pipe diameter and material data. This works in the same fashion as the standard Pipe Segment unit operation.

The external diameter is not currently used by the calculations. It has been added so that the heat transfer models can be more easily enhanced in future versions.

Heat Transfer Page

A simplified heat transfer model is used that allows you to specify the ambient temperature and an overall heat transfer coefficient.

Figure 7.5

The Ambient Temperature is the bulk ambient temperature, and Overall HTC is the overall heat transfer coefficient based upon the inside diameter of the pipe.

7.1.4 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

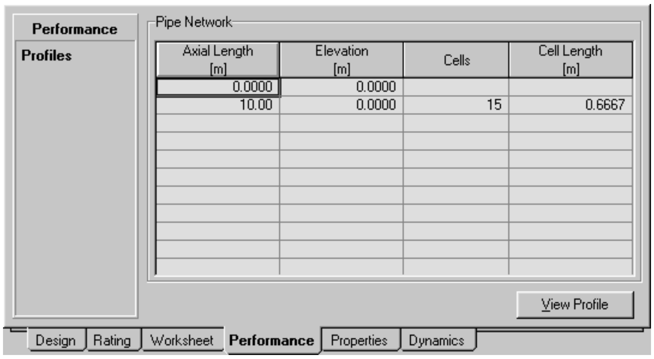
The PF Specs page is relevant to dynamics cases only.

7.1.5 Performance Tab

Refer to **Section 7.3 - Pipe Segment** for more information.

This tab is functionally similar to the Performance tab on the standard Pipe Segment unit operation.

Figure 7.6



You can view a complete profile by clicking on the **View Profile** button. The properties displayed on the Table tab of the Profile property view are listed below:

- Axial Length
- Pressure
- Temperature
- Mass Flow
- Velocity
- Mach Number
- Mass Density
- Internal Energy
- Enthalpy
- Speed Of Sound

7.1.6 Properties Tab

Due to the number of physical property calculations, an acceptable calculation speed is not possible by directly calling the current property package for the flowsheet. Three alternative methods are available from the drop-down list:

- Perfect Gas
- Compressible Gas
- Table Interpolation

The methods are described in the sections below.

Perfect Gas

$$H = C_p \Delta T \tag{7.6}$$

$$\rho = \frac{PMW}{RT} \tag{7.7}$$

Compressible Gas

Same as for perfect gas, but

$$\rho = \frac{PMW}{ZRT} \tag{7.8}$$

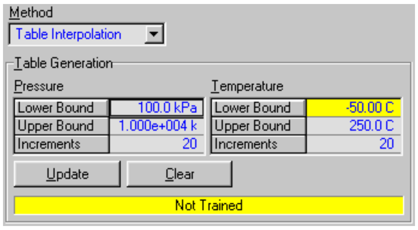
The compressibility factor, Z is calculated from the current property package for the flow sheet at the average conditions within the pipe.

Table Interpolation

A neural network calculates physical properties. This neural network uses a Radial Basis Function to train the network from physical properties, predicted from the current property package of the flowsheet.

Prior to calculations, you must train the neural network. The Table Generation group manages the extent of the training.

Figure 7.7



Care must be taken to train over the full extent of the expected range of operating conditions since extrapolation always yield unpredictable results.

7.1.7 Dynamics Tab

The Dynamics tab contains the following pages:

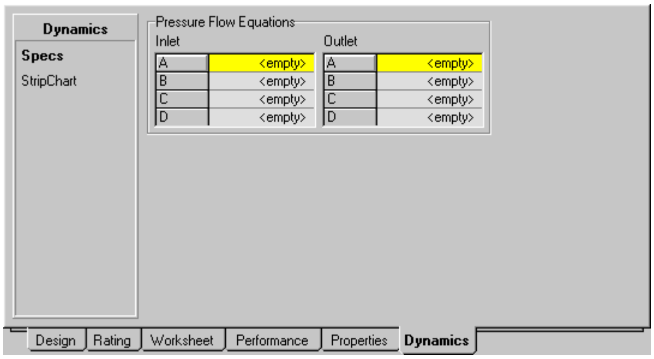
- Specs
- StripChart

Specs Page

For transient compressible flow calculations, the solution of pressure/flow equations is inappropriate since the boundary pressure is not directly related to flow. It is however critical that the compressible gas solve simultaneously with the other flowsheet equations. This is achieved by making perturbations at each end of the pipe for each time step and re-evaluating the change in state over the time step. These changes are then fit to an equation of the following form, which is passed to the Pressure Flow solver:

$$A.\mathrm{Pres} + BFlow^2 + CFlow + D = 0 \tag{7.9}$$

Figure 7.8



The Pressure Flow Equations group displays the values for the coefficients in the above equation, which are continuously updated at each time step.

Stripchart Page

Refer to **Section 1.3.7 - Stripchart Page/Tab** for more information.

The Stripchart page allows you to select and create default strip charts containing various variable associated to the operation.

7.2 Mixer

The Mixer operation combines two or more inlet streams to produce a single outlet stream. A complete heat and material balance is performed with the Mixer. That is, the one unknown temperature among the inlet and outlet streams is always calculated rigorously. If the properties of all the inlet streams to the Mixer are known (temperature, pressure, and composition), the properties of the outlet stream is calculated automatically since the composition, pressure, and enthalpy is known for that stream.

The mixture pressure and temperature are usually the unknowns to be determined. However, the Mixer also calculates backwards and determine the missing temperature for one of the inlet streams if the outlet is completely defined. In this latter case, the pressure must be known for all streams.

The resultant temperature of the mixed streams may be quite different than those of the feed streams due to mixing effects.

The Mixer flashes the outlet stream using the combined enthalpy. Notice that when the inlet streams are completely known, no additional information needs to be specified for the outlet stream. The problem is completely defined; no degrees of freedom remain.

The dynamic Mixer operation functions very similarly to the steady state Mixer operation. However, the enhanced holdup model and the concept of nozzle efficiencies can be applied to the dynamic Mixer. Flow reversal is also possible in the Mixer depending on the pressure-flow conditions of the surrounding unit operations.

7.2.1 Mixer Property View

There are two ways that you can add a Mixer to your simulation:

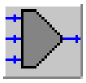
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Piping Equipment** radio button.
3. From the list of available unit operations, select **Mixer**.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

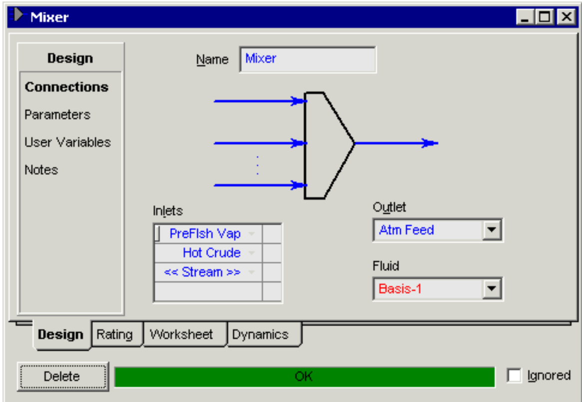
   You can also open the Object Palette by pressing **F4**.
2. Double-click the **Mixer** icon.



Mixer icon

The Mixer property view appears.

Figure 7.9



7.2.2 Design Tab

The Design tab provides access to the following pages:

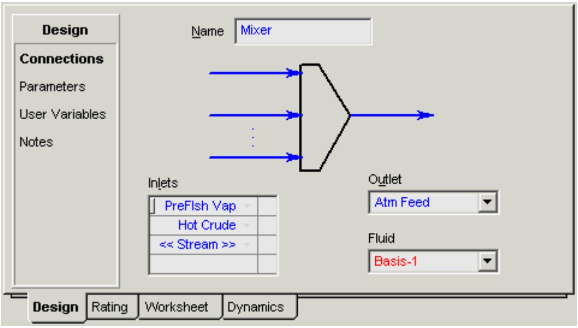
- Connections
- Parameters
- User Variables
- Notes

Connections Page

On the Connections page, you can specify the following:

- any number of inlet streams to the mixer
- a single outlet stream
- name for the mixer
- fluid package associated to the mixer

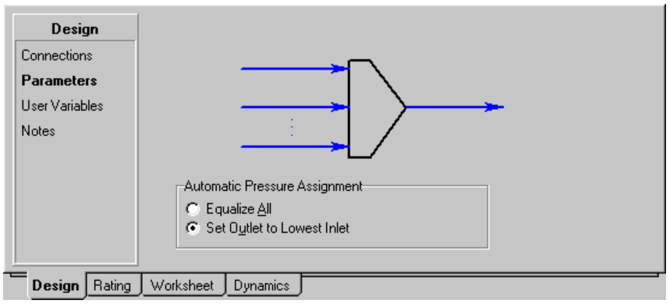
Figure 7.10



Parameters Page

The Parameters page allows you to indicate the type of Automatic Pressure Assignment, HYSYS should use for the streams attached to the Mixer.

Figure 7.11



The default is Set Outlet to Lowest Inlet, in which case all but one attached stream pressure must be known. HYSYS assigns the lowest inlet pressure to the outlet stream pressure.

If you specify Equalize All, HYSYS gives all attached streams the same pressure once one of the attached stream pressures are known. If you want to specify all of the inlet stream pressures, ensure first that all pressures have been specified before installing the Mixer, then choose Set Outlet to Lowest Inlet. In this case, there is no automatic pressure assignment since all the stream pressures are known.

If you select Equalize All and two or more of the attached streams have different pressures, a pressure inconsistency message appears.

In this case, you must either remove the pressure specifications for all but one of the attached streams, or select Set Outlet to Lowest Inlet. If you specify Set Outlet to Lowest Inlet, you can still set the pressures of all the streams.

If you are uncertain of which pressure assignment to use, choose Set Outlet to Lowest Inlet. Only use Equalize All if you are completely sure that all the attached streams should have the same pressure. While the pressure assignment seems to be extraneous, it is of special importance when the Mixer is being used to simulate the junction of multiple pipe nodes.

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

7.2.3 Rating Tab

You need HYSYS dynamics to specify any rating information for the Mixer operation. The Rating tab consists of the Nozzles page.

Nozzles Page

Refer to **Section 1.3.6 - Nozzles Page** for more information.

The Nozzles page contains information regarding the elevation and diameter of the nozzles.

It is strongly recommended that the elevation of the inlet and exit nozzles are equal for this unit operation. If you want to model static head, the entire piece of equipment can be moved by modifying the Base Elevation relative to Ground Elevation field.

7.2.4 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

The PF Specs page is relevant to dynamics cases only.

7.2.5 Dynamics Tab

The Dynamics tab contains the following pages:

- Specs
- Holdup
- Stripchart

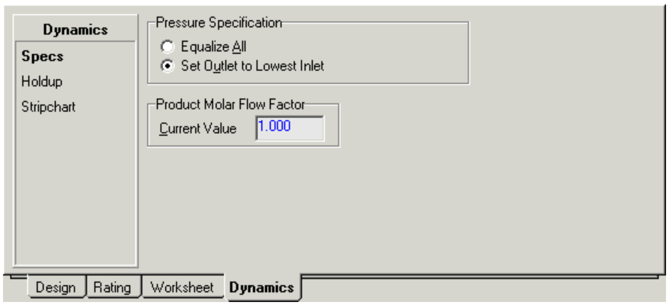
In Dynamic mode, changes in inlet streams to the Mixer are seen instantaneously in the outlet stream because the Mixer is assumed to have no holdup.

If you are working exclusively in Steady State mode, you are not required to change any information on the pages accessible through the Dynamics tab.

Specs Page

The dynamic specifications of the Mixer can be specified on the Specs page.

Figure 7.12



In dynamic mode, there are two possible dynamic specifications you can choose to characterize the Mixer operation:

- If you select the **Equalize All** radio button, the pressure of the surrounding streams of the Mixer are equal if static head contributions are not considered. This is a realistic situation since the inlet stream pressures to a Mixer in an actual plant must be the same. With this specification, flow to and from the Mixer is determined by the pressure flow network. The “one PF specification per flowsheet boundary stream” rule applies to the Mixer operation if the Equalize All option is chosen. It is strongly recommended that you use the Equalize All option in order to realistically model flow behaviour in a dynamic simulation case.
- If you select the **Set Outlet to Lowest Inlet** radio button, HYSYS sets the pressure of the exit stream of the Mixer to the lowest inlet stream pressure. This situation is not recommended since two or more streams can enter the Mixer at different pressures which is not realistic. With this specification, flow to and from the Mixer is determined from upstream flow specifications, and not from the surrounding pressure network of the simulation case. If this option is used, *n* more pressure-

flow specifications are required by the PF solver than if the Equalize All option is used. The variable, n, is the number of inlet streams to the Mixer.

Reverse flow conditions can occur in the Mixer operation if the Equalize All radio button is not selected. If flow reverses in the Mixer, the Mixer essentially acts like a dynamic Tee with the Use Splits as Dynamic Specs checkbox inactive. In dynamics, these two unit operations are very similar.

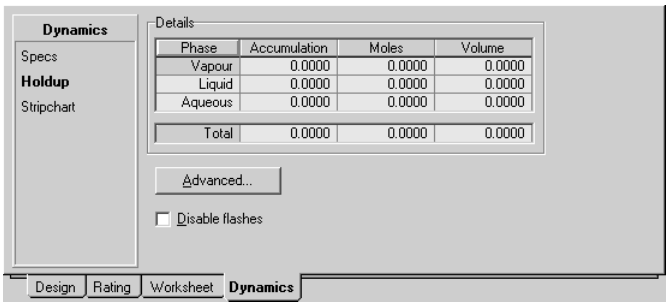
The Product Molar Flow Factor field enables you to scale the flow rate coming out of the mixer. For example, there are two parallel trains but you only want to model one train. You can accomplish modeling one train by changing the Product Molar Flow Factor value, so that the flow rate out of the mixer equals the flow rate value into the mixer multiplied by the Product Molar Flow Factor value.

Holdup Page

Refer to **Section 1.3.3 - Holdup Page** for more information.

Each unit operation in HYSYS has the capacity to store material and energy. Typical Mixers in actual plants usually have significantly less holdup than other unit operations in a plant. Therefore, the volume of the Mixer operation in HYSYS cannot be specified and is assumed to be zero. Since there is no holdup associated with the Mixer operation, the holdup's quantity and volume are shown as zero in the Holdup page.

Figure 7.13



The **Disable flashes** checkbox enables you to turn on and off the rigorous flash calculation for the mixer. This feature is useful if the PFD has a very large number of mixers, and you do not care whether the contents of the streams around them are fully up to date or not, or you prefer maximum speed in the simulation calculation.

- To turn off the flash calculation, select the **Disable flashes** checkbox.

  If the flash calculations are turned off, the outlet stream will still update and propagate values, but the phase fractions and temperatures may not be correct.
- To turn the flash calculation back on, clear the **Disable flashes** checkbox.

The default selection is to leave the flash calculation on.

HYSYS recommend that the flash calculations be left on, as in some cases disabled flash calculation can result in instabilities or unexpected outcomes, depending on what is downstream of the unit operation where the flash has been turned off. This feature should only be manipulated by advanced users.

Stripchart Page

Refer to **Section 1.3.7 - Stripchart Page / Tab** for more information.

The Stripchart page allows you to select and create default strip charts containing various variable associated to the operation.

7.3 Pipe Segment

The Pipe Segment is used to simulate a wide variety of piping situations ranging from single or multiphase plant piping with rigorous heat transfer estimation, to a large capacity looped pipeline problems. It offers several pressure drop correlations:

- Aziz, Govier, and Fogarasi
- Baxendell and Thomas
- Beggs and Brill
- Duns and Ros
- Gregory Aziz Mandhane
- Hagedorn and Brown
- HTFS, Liquid Slip
- HTFS, Homogeneous Flow

- OLGAS2000_2P
- OLGAS2000_3P
- Orkiszewski
- Poettmann and Carpenter
- Tacite Hydrodynamic Module
- Tulsa 99

Another option, **OLGAS**, is also available as a gradient method. Four levels of complexity in heat transfer estimation allow you to find a solution as rigorous as required while allowing for quick generalized solutions to well-known problems.

Contact your AspenTech agent for more information, or e-mail us at **info@aspentech.com**.

OLGAS is a third-party option which can be purchased through AspenTech or SCANDPOWER.

The Pipe Segment offers four calculation modes. The appropriate mode is automatically selected depending on the information specified. In order to solve the pipe, you must specify enough information to completely define both the material balance and energy balance.

Calculation Modes

The Pipe Segment operation contains four calculation modes:

- Pressure Drop
- Length
- Flow
- Diameter

The mode is automatically assigned depending on what information is specified.

HYSYS checks for sonic flow if indicated by the option in the Calculation page of the Design tab.

Regardless of which mode you use, you must specify the number of increments in the pipe. Calculations are performed in each increment; for example, to determine the pressure drop, the energy and mass balances calculations are performed in each increment, and the outlet pressure in that increment is used as the inlet pressure to the next increment. The calculation continues down the length of the pipe until the pipe outlet pressure is determined.

The Pipe Segment can solve in either direction. The solution procedure generally starts at the end where the temperature is known (temperature is typically not known on both ends). HYSYS then begins stepping through the pipe from that point, using either the specified pressure, or estimating a starting value. If the starting point is the pipe outlet, HYSYS steps backwards through the pipe. At the other end of the pipe, HYSYS compares the calculated solution to other known information and specifications, and if necessary, restarts the procedure with a new set of starting estimates.

Some specifics of each calculation mode are provided in the following sections.

Pressure Drop

Assuming that a feed, product, and energy stream are attached to the pipe, the following information is required:

- Flow
- Pipe length, diameter, and elevation change
- Heat transfer information
- At least one stream temperature and one pressure

There are two different methods for calculating the pressure drop, which are discussed below:

Method 1

If you specify the temperature and pressure at the same end of the pipe, then energy and mass balances are solved for each increment, and the temperature and pressure of the stream at the opposite end of the pipe are determined.

Delta P Method 1:

1. At the end where temperature and pressure are specified, solve for the outlet temperature and pressure in the first segment.
2. Move to the next segment, using the outlet conditions of the previous segment as the new inlet conditions.
3. Continue down the pipe until the outlet pressure and temperature are solved.

Method 2

If you specify temperature for one stream and pressure for the other, an iterative loop is required outside of the normal calculation procedure:

- First, a pressure is estimated for the stream which has the temperature specified.
- Second, the pressure and temperature for the stream at the opposite end of the pipe are determined from incremental energy and mass balances as in the first method.
- If the calculated pressure and user-specified pressure are not the same (within a certain tolerance), a new pressure is estimated and the incremental energy and mass balances are re-solved. This continues until the absolute difference of the calculated and user-specified pressures are less than a certain tolerance.

The calculated pressure drop accounts for fittings, frictional, and hydrostatic effects.

Delta P Method 2:

1. Estimate a pressure for the stream which has a specified temperature.
2. At the end where the pressure is estimated, solve for the outlet temperature and pressure in the first segment.
3. Move to the next segment, using the outlet conditions of the previous segment as the new inlet conditions.
4. Continue down the pipe until the outlet pressure and temperature are solved.
5. If the calculated outlet pressure is not equal to the actual pressure, a new estimate is made for pressure (Return to 1).

Length

Assuming that the feed, product, and energy stream are attached, the following information is required:

- Flow
- Heat transfer information
- Pipe diameter
- Inlet and Outlet Pressure (or one stream Pressure and Pressure Drop)
- One stream temperature
- Initial estimate of Length

For each segment, the Length estimate, along with the known stream specifications, are used to solve for the unknown stream temperature and pressure. If the calculated pressure is not equal to the actual pressure (within the user-specified tolerance), a new estimate is made for the length, and calculations continue.

A good initial guess and step size greatly decreases the solving time.

The Pipe also solves for the length if you provide one pressure, two temperature specifications, and the duty.

Length Calculation:

1. Estimate a Length. At the end where temperature is specified, solve for the outlet temperature and pressure in the first segment.
2. Move to the next segment, using the outlet conditions of the previous segment as the new inlet conditions.
3. Continue down the pipe until the outlet pressure and temperature are solved.
4. If the calculated outlet pressure is not equal to the actual pressure, a new estimate is made for length. (Return to 1).

Diameter

Information required in the Diameter calculation mode is the same as Length, except HYSYS requires the length instead of the diameter of the pipe. Initial estimate of diameter can be given on the Calculation page of the Design tab.

Both length and diameter calculations can only be done for pipes with a single segment.

Flow

Assuming that a feed, product, and energy stream are attached to the pipe, the following information is required:

- Pipe length and diameter
- Heat transfer information
- Inlet and Outlet Pressure (or one stream Pressure and Pressure Drop)
- One stream temperature
- Initial estimate of Flow

Using the flow estimate and known stream conditions (at the end with the known temperature), HYSYS calculates a pressure at the other end. If the calculated pressure is not equal to the actual pressure (within the user-specified tolerance), a new estimate is made for the flow, and calculations continue. Again, a good initial guess decreases the solving time significantly.

Flow Calculation:

1. Estimate Flow. At the end where temperature is specified, solve for the outlet temperature and pressure in the first segment.
2. Move to the next segment, using the outlet conditions of the previous segment as the inlet conditions.
3. Continue down the pipe until the outlet pressure and temperature are solved.
4. If the calculated outlet pressure is not equal to the actual pressure, a new estimate is made for the flow. (Return to 1).

Incremental Material and Energy Balances

The overall algorithm consists of three nested loops. The outer loop iterates on the increments (Pressure, Length or Flow Mode), the middle loop solves for the temperature, and the inner loop solves for pressure. The middle and inner loops implement a secant method to speed convergence.

The pressure and temperature are calculated as follows:

1. The inlet temperature and pressure are passed to the material/energy balance routine.
2. Using internal estimates for temperature and pressure gradients, the outlet temperature and pressure are calculated.
3. Average fluid properties are calculated based on the inlet and estimated outlet conditions.
4. These properties, along with the inlet pressure, are passed to the pressure gradient algorithm.
5. With the pressure gradient, the outlet pressure can be calculated.
6. The calculated pressure and estimate pressure are compared. If their difference exceeds the tolerance (default value 0.1 kPa), a new outlet pressure is estimated, and steps #3 to #6 are repeated.

   The tolerance is specified in the Calculation page of the Design tab.
7. Once the inner pressure loop has converged, the outlet temperature is calculated:
   - If U and the ambient temperature are specified, then the outlet temperature is determined from the following equations:

$$Q = U \times A \times \Delta T_{LM} \tag{7.10}$$

$$Q = Q_{in} - Q_{out} \tag{7.11}$$

where:

Q = amount of heat transferred

U = overall heat transfer coefficient

A = outer heat transfer area

ΔT_{LM} *= log mean temperature difference*

Q_{in} *= heat flow of inlet stream*

Q_{out} *= heat flow of outlet stream*

- If both the inlet and outlet Pipe temperatures are known, the outlet temperature of the increment is calculated by linear interpolation. The attached duty stream then completes the energy balance.
- If duty is known, the outlet temperature is calculated from a Pressure-Enthalpy flash.

When the Increment outlet temperature is calculated, it is compared with the estimated outlet temperature. If their difference exceeds the tolerance (default value 0.01°C), a new outlet temperature is estimated, and new fluid properties are calculated (return to step #3). The tolerance is specified in the Calculation page of the Design tab.

8. When both the temperature and pressure converge, the outlet results are passed to the inlet of the next increment, where calculations continue.

7.3.1 Pipe Segment Property View

There are two methods to add a Pipe Segment to the simulation:

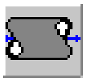
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Piping Equipment** radio button.
3. From the list of available unit operations, select **Pipe Segment**.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.

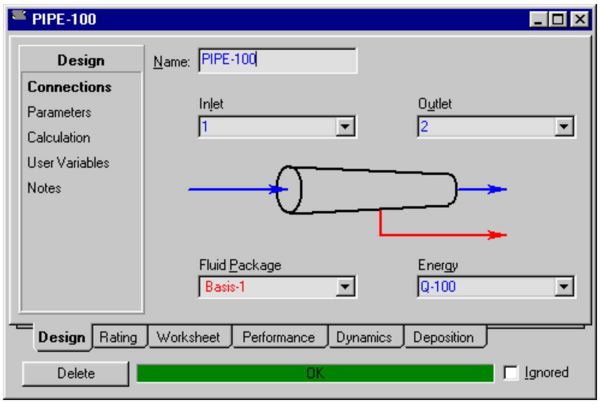


Pipe Segment icon

2. Double-click the **Pipe Segment** icon.

The Pipe Segment property view appears.

Figure 7.14



7.3.2 Design Tab

The Design tab contains the following pages:

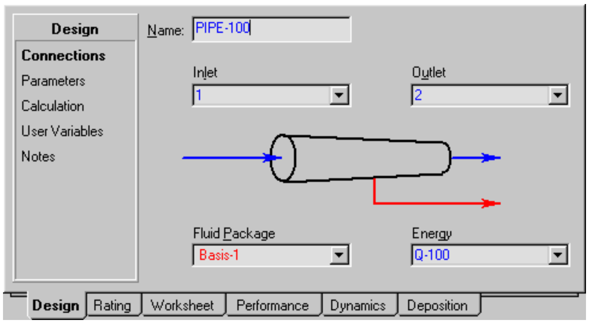
- Connections
- Parameters
- Calculation
- User Variables
- Notes

Connections Page

On the Connections page, you must specify the feed and product material streams.

In the Inlet, Outlet and Energy drop-down lists either type in the name of the stream or if you have pre-defined your stream select it from the drop-down list.

Figure 7.15

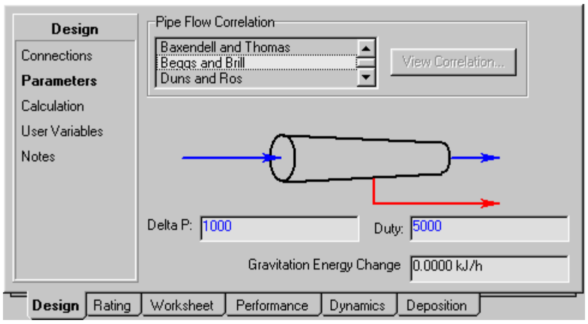


In addition to the material stream connections, you also have the option of attaching an energy stream to the Pipe Segment and selecting the fluid package for the Pipe Segment. You can also edit the Pipe Segment name on this page.

Parameters Page

In the Pipe Flow Correlation group, you can select the correlation method used for Two Phase (VL) flow calculations.

Figure 7.16



The options are:

- Aziz, Govier, and Fogarasi
- Baxendell and Thomas
- Beggs and Brill
- Duns and Ros
- Gregory Aziz Mandhane

- Hagedorn and Brown
- HTFS, Liquid Slip
- HTFS, Homogeneous Flow
- OLGAS2000_2P

OLGAS is a third-party option that can be purchased through AspenTech or SCANDPOWER.

- OLGAS2000_3P
- Orkiszewski
- Poettmann and Carpenter
- Tacite Hydrodynamic Module
- Tulsa 99

Summary of Methods

The methods above have all been developed for predicting two-phase pressure drops. Some methods were developed exclusively for flow in horizontal pipes, others exclusively for flow in vertical pipes while some can be used for either. Some of the methods define a flow regime map and can apply specific pressure drop correlations according to the type of flow predicted. Some of the methods calculate the expected liquid holdup in two-phase flow while others assume a homogeneous mixture.

The table below summarizes the characteristics of each model. More detailed information on each model is presented later in this section.

Model	Horizontal Flow	Vertical Flow	Liquid Holdup	Flow Map
Aziz, Govier & Fogarasi	No	Yes	Yes	Yes
Baxendell & Thomas	Use with Care	Yes	No	No
Beggs & Brill	Yes	Yes	Yes	Yes
Duns & Ros	No	Yes	Yes	Yes
Gregory, Aziz, Mandhane	Yes	No	Yes	Yes
Hagedorn & Brown	No	Yes	Yes	No
HTFS Homogeneous	Yes	Yes	No	No
HTFS Liquid Slip	Yes	Yes	Yes	No

Model	Horizontal Flow	Vertical Flow	Liquid Holdup	Flow Map
Olgas2000	Yes	Yes	Yes	Yes
Orkisewski	No	Yes	Yes	Yes
Poettman & Carpenter	No	Yes	No	No
Tacite Hydrodynamic Module	Yes	Yes	Yes	Yes
Tulsa	No	Yes	Yes	Yes

For Single Phase streams, the Darcy equation is used for pressure drop predictions. This equation is a modified form of the mechanical energy equation, which takes into account losses due to frictional effects as well as changes in potential energy.

The total heat loss from the Pipe Segment is indicated in the Duty field. The total heat loss can be calculated using estimated heat transfer coefficients or specified on the Heat Transfer page of the Rating tab.

You can also specify the overall pressure drop for the operation. The pressure drop includes the losses due to friction, static head, and fittings. If the overall pressure drop is not specified on the Parameters page, it is calculated by HYSYS, provided all other required parameters are specified.

The overall pressure drop, which can be specified or calculated by HYSYS, is the sum of the friction, static head, and fittings pressure drops.

When two liquid phases are present, appropriate volume based empirical mixing rules are implemented to calculate a single pseudo liquid phase. Therefore, caution should be exercised in interpreting the calculated pressure drops for three-phase systems.

Actual pressure drops can vary dramatically for different flow regimes, and for emulsion systems.

The Gravitational Energy Change field displays the change in potential energy experienced by the fluid across the length of the pipe. It is determined for the overall elevation change, based on the sum of the elevation change specified for each segment on the Sizing page of the Rating tab.

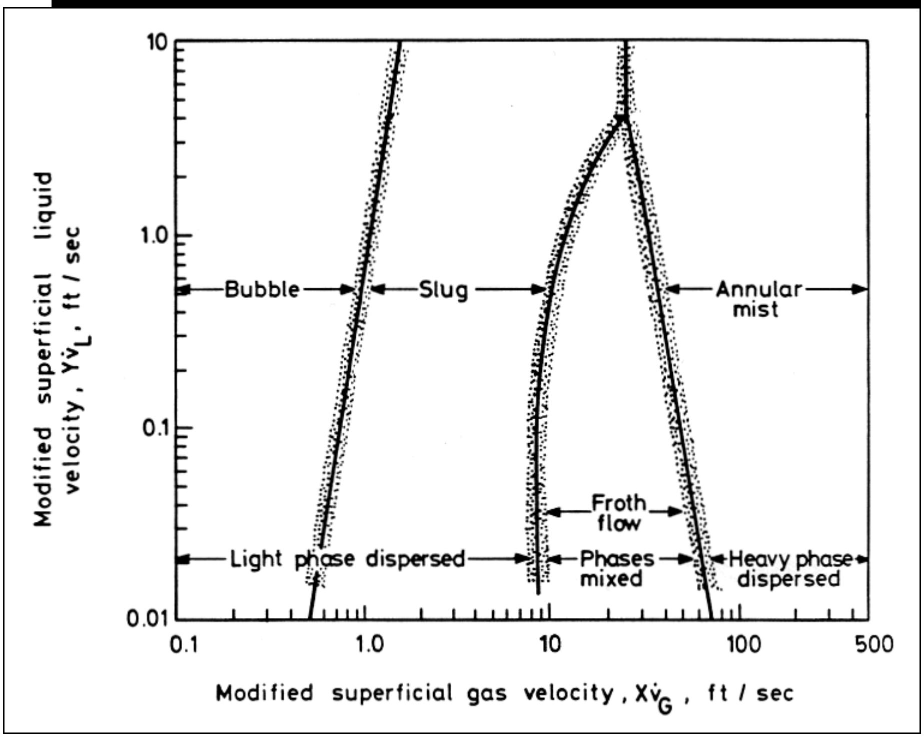
When the pressure drop is specified, the Pipe Segment can be used to calculate either the length of the Pipe Segment or the flow of the material through the length of pipe.

Notice the calculation type (for example, pressure drop, length, flow) is not explicitly specified. HYSYS determines what is to be calculated by the information that you provide.

Aziz, Govier & Fogarasi

In developing their model[2] Aziz, Govier & Fogarasi argue that flow regime is independent of phase viscosities and pipe diameters but is proportional to the gas density to the one third power ($\rho_g^{1/3}$). From this, then the calculate modified superficial gas and liquid velocities on which they base the following flow regime map.

Figure 7.17



Once the flow regime has been determined a range of correlations is used to determine the frictional pressure gradient and slip velocity or void fraction applicable to that regime.

Baxendell & Thomas

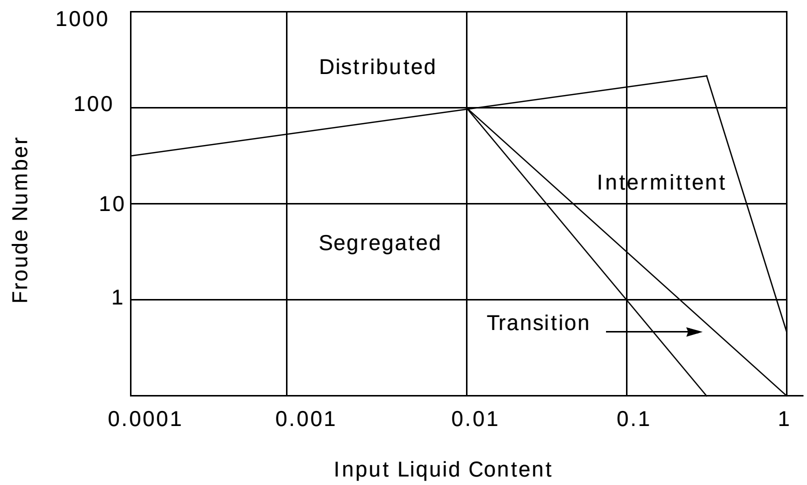
The Baxendell & Thomas model[3] is an extension of the Poettman & Carpenter model to include higher flow rates. It is based on a homogeneous model using a two-phase friction factor obtained from correlation based on experimental results relating friction factor to the parameter $D\rho\upsilon$. Baxendell & Thomas fitted a smooth curve for values of the $D\rho\upsilon$ parameter greater than 45 x103 cp. Below this value they propose that original correlation of Poettman & Carpenter be used. Baxendell & Thomas claim the correlation is suitable for use in calculating horizontal flow pressure gradients in addition to the vertical flow pressure gradients for which the original Poettman & Carpenter approach was developed although the correlation takes no account of the very different flow regimes that can occur. Like the Poettman & Carpenter model this model assumes that the pressure gradient is independent of viscosity.

Beggs and Brill Pressure Gradient

The Beggs and Brill[4] method is based on work done with an air-water mixture at many different conditions, and is applicable for inclined flow.

Figure 7.18

Beggs and Brill Flow Regimes



In the Beggs and Brill correlation, the flow regime is determined using the Froude number and inlet liquid content. The flow map used is based on horizontal flow and has four regimes: segregated, intermittent, distributed, and transition. The types of flow in the first three regime are listed as follows:

- **Segregated Flow**: Stratified, Wavy, and Annular.
- **Intermittent Flow**: Plug and Slug.
- **Distributed Flow**: Bubble and Mist.

Once the flow regime has been determined, the liquid holdup for a horizontal pipe is calculated, using the correlation applicable to that regime. A factor is applied to this holdup to account for pipe inclination. From the holdup, a two-phase friction factor is calculated and the pressure gradient determined.

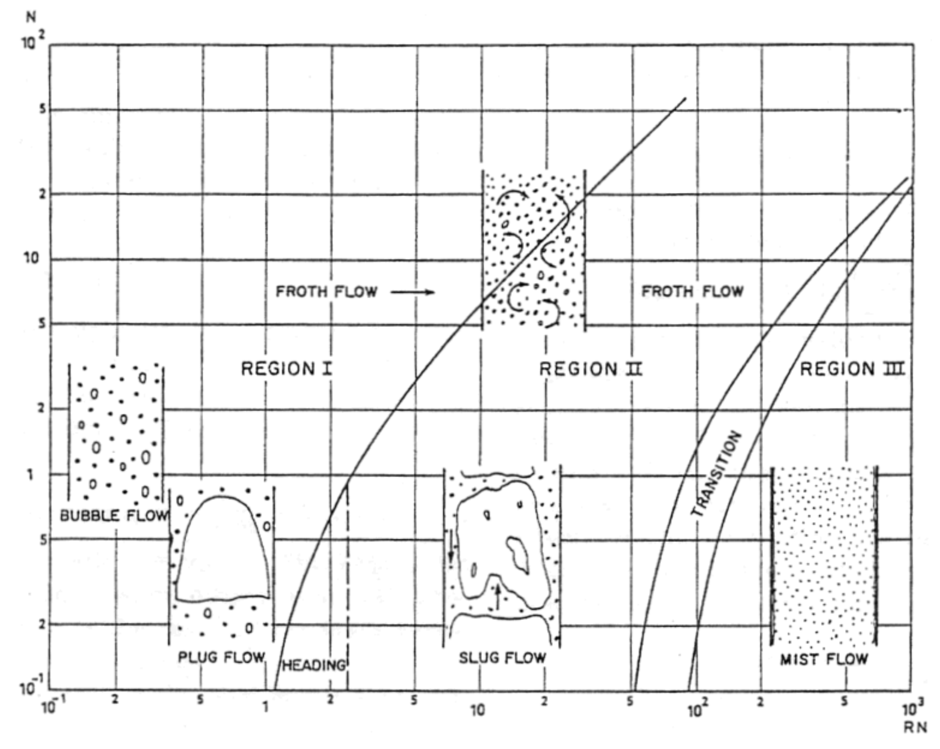
Duns & Ros

The Duns and Ros model[8] is based on a large scale laboratory investigation of upward vertical flow of air / hydrocarbon liquid and air / water systems. The model identifies three flow regions, outlined below.

- **Region I**. Where the liquid phase is continuous (in other words, bubble and plug flow, and part of froth flow regimes).
- **Region II**. Where the phases of liquid and gas alternate (in other words, remainder of froth flow regime and slug flow regime).
- **Region III**. Where gas phase is continuous (in other words, mist flow and annular flow regime).

The flow region map is shown in the figure below:

Figure 7.19



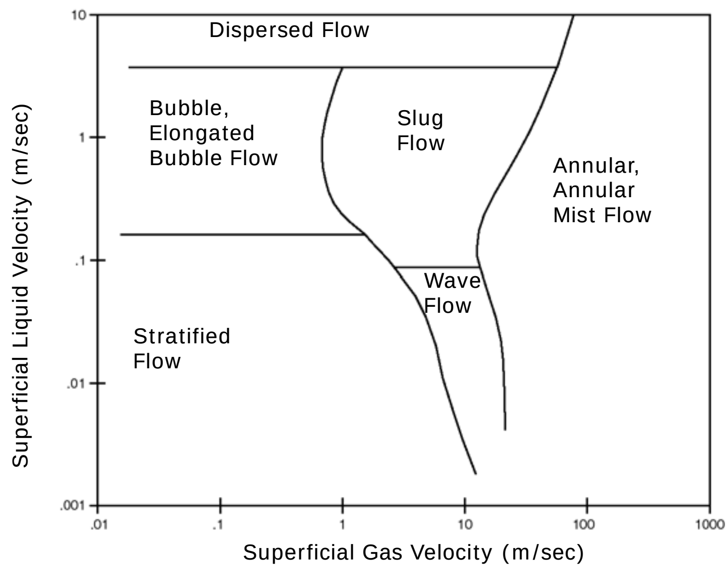
The regions are distinguished using functions of four dimensionless groups namely a gas velocity number, a liquid velocity number, a diameter number, and a liquid viscosity number. Separate frictional pressure drop correlations and liquid slip velocity (liquid holdup) correlations are defined for each region in terms of the same dimensionless groups.

Gregory Aziz Mandhane Pressure Gradient

For the Gregory Aziz Mandhane correlation[10], an appropriate model is used for predicting the overall pressure drop in two-phase flow.

Figure 7.20

Gregory Aziz Mandhane Flow Regimes



Regime	Model
Slugflow	Mandhane, et. al. modification #1 of Lockhart-Martinelli
Dispersed	Bubble Mandhane, et. al. modification #2 of Lockhart-Martinelli
Annular Mist	Lockhart-Martinelli
Elongated Bubble	Mandhane, et. al. modification #1 of Lockhart-Martinelli

Regime	Model
Stratified	Lockhart-Martinelli
Wave	Lockhart-Martinelli

Hagedorn & Brown

Hagedorn & Brown based their model[11] on experimental data on upward flow of air / water and air / oil mixtures. The frictional pressure drop is calculated using a friction factor derived from a single phase Moody curve using a two phase Reynolds number that reduces to the appropriate single phase Reynolds number when the flow becomes single phase. For the void fraction required to calculate the two phase Reynolds number and the static pressure loss, Hagedorn & Brown developed a single curve relating the void fraction to the same dimensionless parameters proposed by Duns & Ros.

HTFS Models

The two HTFS models[12, 17] share a common method for calculating the frictional pressure gradient and acceleration pressure gradient while differing in the method used to calculate static pressure gradient.

The frictional pressure gradient method is adapted from that of Claxton et. al. (1972). The method first calculates the frictional pressure drop for the gas and liquid phases assuming that they are flowing alone in the pipe based on Fanning friction factors for each phase that are again calculated by assuming the fluid is flowing alone in the pipe. The frictional pressure drop is then calculated from the formula:

$$\Delta p_F = \Delta p_l + C_c\sqrt{(\Delta p_l \Delta p_g)} + \Delta p_g \tag{7.12}$$

where:

Δp_F = *frictional pressure drop*

Δp_l = *liquid phase pressure drop*

C_c = correction factor calculated from the properties of the liquid and gas phases and the superficial mass velocities of the phases

Δp_g = gas phase pressure drop

The static pressure gradient is calculated from a separated model of two phase flow. In the HTFS Homogeneous model the void fraction required by this model is assumed to be the homogeneous void fraction. In the HTFS Liquid Slip model the void fraction is calculated using a method published by Whalley and Ward (1981).

The accelerational gradient term is calculated from a homogeneous equation model.

The HTFS models have been validated for horizontal, and both upward and downward vertical flow using a wide range of data held by the Harwell data bank.

OLGAS2000 (2-Phase & 3-Phase)

OLGAS2000 employs mechanistic models for each of the four major flow regimes: stratified, annular, slug, and dispersed bubble flow. It is based in large part on data from the SINTEF multiphase flow laboratory in Norway.

Multiphase Flow is a dynamic physical process between the phases. It includes fluid properties, complex geometry and interaction between reservoir, well, flowline and process plant. OLGAS 2000 can handle 2-phase and 3-phase flow. For instance, the elements involved can consist of water droplets, oil, gas, sand, wax, and hydrates.

OLGAS2000 predicts the pressure gradient, liquid holdup, and flow regime. It has been tested in one degree increments for all angles from horizontal to vertical. OLGAS2000 gives one of the best overall predictions of pressure drop and liquid holdup of any currently available method.

Contact your AspenTech agent for more information on OLGAS2000 and the licensing on OLGAS2000 3-Phase.

Orkisewski

Orkisewski[15] composed a composite correlation for vertical upward flow based on a combination of methods developed by Griffith (1962), Griffith & Wallis(1961), and Duns & Ros (1963)[8]. Four flow regimes are defined and the methods proposed for each region are:

- Bubble flow—Griffith correlation
- Slug/Plug flow—Griffith & Wallis correlation modified by Orkisewski
- Churn flow—Duns & Ros
- Mist/Annular flow—Duns & Ros

Orkisewski proposed that the method of Griffith and Wallis be used to determine the boundary between the bubble and plug flow regime and the methods of Duns & Ros be used to determine the remaining flow regime boundaries.

Poettman & Carpenter

The Poettman & Carpenter model[16] assumes that the contribution of the acceleration term to the total pressure loss is small and that the frictional pressure drop can be calculated using a homogeneous model. The model further assumes that the static head loss can be calculated using a homogeneous two phase density. Poettman & Carpenter varies from a standard homogeneous method in its calculation of a two phase friction factor. The model proposes a correlation for the friction factor based on experimental results from 49 flowing and gas lift wells operating over a wide range of conditions. The two-phase friction factor is plotted against the parameter $D\rho\upsilon$ (D= diameter, ρ = homogeneous density, and υ = homogeneous superficial velocity). Effectively therefore the model assumes that the pressure gradient is independent of viscosity.

Tacite Hydrodynamic Module

The Tacite Hydrodynamic Module is a transient multi-component two-phase flow simulator for the design and control of oil and gas pipelines. This module provides two modelling options, full gas-liquid modeling and Zuber-Findlay, for predicting the flow behaviour, pressure drop, Barycentric velocity, gas slug fraction, frictional heat transfer coefficient, and volume fraction of a fluid in a horizontal, or inclined pipeline.

The Tacite Hydrodynamic Module is designed for two-phase flow computation, thus water and the oil phase are defined as the same liquid phase.

The model identifies three flow patterns: stratified, intermittent, and dispersed.

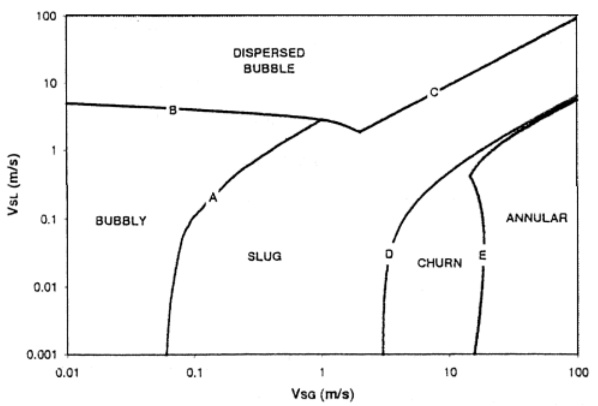
- **Stratified flow**. The model assumes a momentum balance between the phases present in the pipe segment.
- **Intermittent flow**. The intermittent flow regime is solved as a two-region problem. The gas pocket is considered as a stratified flow, whereas the liquid slug is considered as the dispersed flow. The Tacite Hydrodynamic Module can predict the propagation of liquid slug flow that occurs during transient flow conditions in a pipeline. Liquid slugs are created during flow rate changes, pipeline depressurization, shutdown, and startup operations or variations in pipeline topography. Closure laws are used for calculating the slug velocity and gas fraction in the slug.
- **Dispersed flow**. The regime is a particular case of intermittent flow.

Tulsa

The Tulsa model[18] proposes a comprehensive mechanistic model formulated to predict flow patterns, pressure drop, and liquid holdup in vertical upward two-phase flow. The model identifies five flow patterns: bubble, dispersed bubble, slug, churn, and annular. The flow pattern prediction models used are Ansari et. al. (1994) for dispersed bubble and annular flows, Chokshi (1994) for bubbly flow and a new model for churn flow.

The resulting flow pattern map is shown below.

Figure 7.21



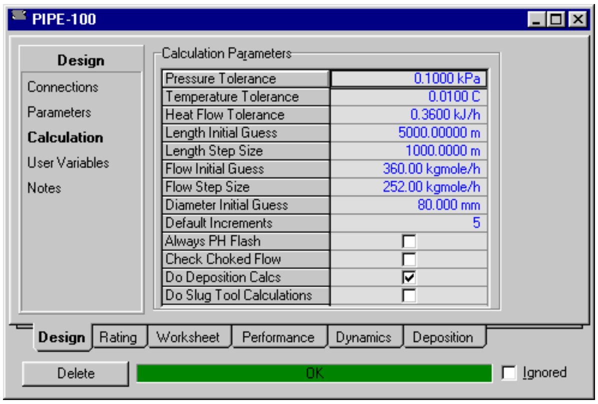
Separate hydrodynamic models for each flow pattern are used. A new hydrodynamic model is proposed for churn flow and a modified version of Chokshi's model is proposed for slug flow. Chokshi and Ansari et. al. models are adopted for bubbly and annular flows respectively.

The model has been evaluated using the Tulsa University Fluid Flow Projects well data back of 2052 wells covering a wide range of field data. The model has been compared with Ansari et. al. (1994), Chokshi (1994), Hasan & Kabir (1994), Aziz et. al. (1972), and Hagedorn and Brown (1964) methods, and is claimed to offer superior results.

All methods account for static head losses, while Aziz, Beggs and Brill, and OLGAS methods account for hydrostatic recovery. Beggs and Brill calculate the hydrostatic recovery as a function of the flow parameters and pipe angle.

Calculation Page

Figure 7.22



You can specify any of the calculation parameters on this page. The table below describes the parameters.

Field	Description
Pressure Tolerance	Tolerance used to compare pressures in the calculation loop.
Temperature Tolerance	Tolerance used to compare temperatures in the calculation loop.
Heat Flow Tolerance	Tolerance used to compare heat flow in the calculation loop.
Length Initial Guess	Used in the algorithm when length is to be calculated.
Length Step Size	Used in the algorithm when length is to be calculated.
Flow Initial Guess	Used in the algorithm when flow of material is to be calculated.
Flow Step Size	Used in the algorithm when flow of material is to be calculated
Diameter Initial Guess	Optional estimate when diameter is to be calculated.
Default Increments	The increment number which appears for each segment on the **Dimensions** page
Always PH Flash	Selecting this checkbox, force HYSYS' calculations to be done using PH flashes rather than PT flashes. Slower but more reliable for pure component or narrow boiling range systems.

Field	Description
Check Choked Flow	When this checkbox is active, HYSYS checks for choked flow. The default setting is inactive because the command slows down calculations. This check is carried out only on pipe segments not on fitting or swage segments.
Do Deposition Calcs	When this checkbox is inactive, HYSYS turns off deposition calculations. This checkbox is a duplicate of the checkbox on the Deposition tab
Do Slug Tool Calculations	When this checkbox is active, HYSYS performs slug calculations.

When calculating Flow or Length, good initial guesses and step sizes can greatly reduce solution time.

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

7.3.3 Rating Tab

The Rating tab provides access to the following pages:

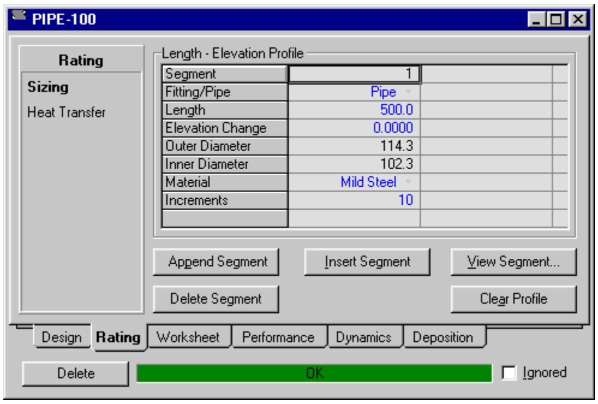
- Sizing
- Heat Transfer

On the Sizing page, you can specify information regarding the dimensions of sections in the Pipe Segment. In the Heat Transfer page, the heat loss of the Pipe Segment can either be specified or calculated from various heat transfer parameters.

Sizing Page

On the Sizing page, the length-elevation profile for the Pipe Segment is constructed. You can provide details for each fitting or pipe section that is contained in the Pipe Segment that you are modeling. An unlimited number of pipe sections or fittings can be added on this page.

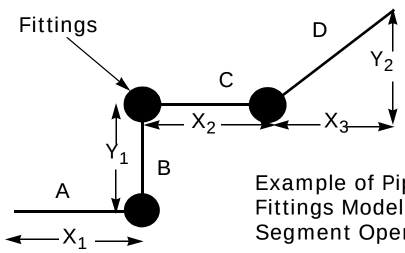
Figure 7.23



For a given length of pipe which is modelled in HYSYS, the parameters of each segment is entered separately, as they are for each fitting.

The procedure for modeling a length of pipe is illustrated using the diagram shown below. In the diagram, the pipe length AD is represented by segments A, B, C, D, and three fittings.

Figure 7.24



Example of Pipe Sections and Fittings Modelled in the Pipe Segment Operation

The table shown below displays the fitting/pipe, length, and elevation input that you require to represent the pipe length AD. Each pipe section and fitting is labelled as a segment.

Number	1	2	3	4	5	6	7
Represented by	A	F1	B	F2	C	F3	D
Fitting/Pipe	Pipe	Fitting	Pipe	Fitting	Pipe	Fitting	Pipe
Length	x_1	N/A	y_1	N/A	x_2	N/A	$\sqrt{x_3^2 + y_2^2}$
Elevation	0	N/A	y_1	N/A	0	N/A	y_2

The horizontal pipe sections have an Elevation of 0. A positive elevation indicates that the outlet is higher than the inlet.

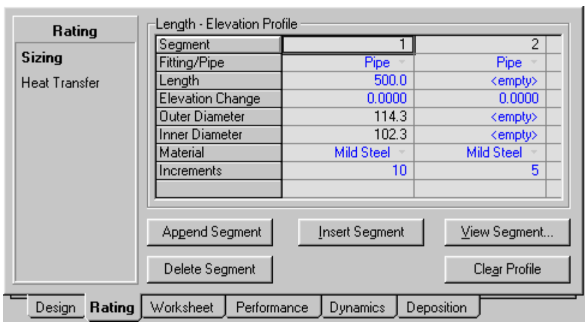
To fully define the pipe section segments, you must also specify pipe schedule, diameters (nominal or inner and outer), a material, and a number of increments. The fittings require an inner diameter value.

When you have only one pipe segment HYSYS calculates the inner diameter of the pipe when a pressure difference and pipe length is specified.

Adding Segments

You can add segments to the length-elevation profile by clicking the Append Segment button.

Figure 7.25



For each segment that you add, you must specify the following:

For more information, refer to **Section 7.3.9 - Modifying the Fittings Database**.

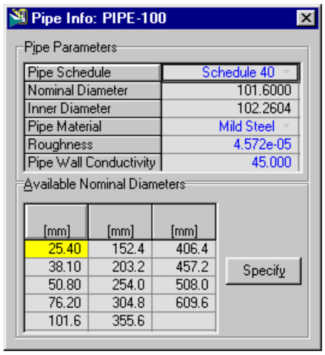
Field	Description
Pipe/Fitting/ Swage	Select a pipe section, swage or one of the available fittings from the drop-down list. If the list does not contain the fitting required, you can modify the fittings and change its K-factor for these calculations. You can modify the Fittings Database, which is contained in file FITTING.DB.
Length	The actual length of the Pipe Segment. Not required for fittings.
Elevation Change	The change in vertical distance between the outlet and inlet of the pipe section. Positive values indicate that the outlet is higher than the inlet. Not required for fittings.
Outer Diameter	Outside diameter of the pipe or fitting.
Inner Diameter	Inside diameter of the pipe or fitting.
Material	Select one of the available default materials or choose User Specified for the pipe section. Not required for fittings.
Increments	The number of increments the pipe section is divided for calculation purposes.

The pipe segment report has been updated to include dedicated detail sections for both fittings and swage fittings. These sections appear in the parameters datablock.

Once you have selected the segment type (pipe, swage, or fitting), you can specify detailed information concerning the highlighted segment. With the cursor located on a segment, click the **View Segment** button. When you click the **View Segment** button, the Pipe Fittings, Pipe Swages, or Pipe Info property view appears. The property view that appears depends on the type of Fitting/Pipe option you selected from the drop-down list.

Viewing Segments

Figure 7.26



The Pipe Info property view appears for pipe sections. On this property view, the following information is shown:.

Field	Description
Pipe Schedule	Select one of the following: • **Actual**. The nominal diameter cannot be specified. The inner diameter can be specified. • **Schedule 40** • **Schedule 80** • **Schedule 160** HYSYS contains a pipe database for three pipe schedules (40, 80, 160). If a schedule is specified, a popup menu appears indicating the possible nominal pipe diameters that can be specified.
Nominal Diameter	Provides the nominal diameter for the pipe section.
Inner Diameter	For Schedule 40, 80, or 160, this is referenced from the database. For Actual Pipe Schedule, this can be specified directly by the user.
Pipe Material	Select a pipe material or choose User Specified. The pipe material type can be selected from the drop-down list in the field. A table of pipe materials and corresponding Absolute Roughness factors is shown in the next table. The roughness factor is automatically specified for pipe material chosen from this list. You can also specify the roughness factor manually.

Field	Description
Roughness	A default value is provided based on the Pipe Material. You can specify a value if you want.
Pipe Wall Conductivity	Thermal conductivity of pipe material in W/m.K to allow calculation of heat transfer resistance of pipe wall. Defaults provided for standard pipe materials are as follows: • All steel and coated iron pipes: 45.0 • Cast iron: 48.0 • Concrete: 1.38 • Wood: 0.173 • PlasticTubing: 0.17 • RubberHose: 0.151

Pipe Material Type	Absolute Roughness, m
Drawn Tube	0.0000015
Mild Steel	0.0000457
Asphalted Iron	0.0001220
Galvanized Iron	0.0001520
Cast Iron	0.0002590
Smooth Concrete	0.0003050
Rough Concrete	0.0030500
Smooth Steel	0.0009140
Rough Steel	0.0091400
Smooth Wood Stave	0.0001830
Rough Wood Stave	0.0009140

Fitting Pressure Loss

The fittings pressure loss is characterised by a two constant equation as shown below.

$$K = A + B \times f_T \tag{7.13}$$

where:

A = constant, also known as velocity head factor

B = constant, also known as FT factor

f_T = fully turbulent friction factor

The fittings pressure loss constant K is then used to obtain the pressure drop across the fitting from the equation shown below.

$$\Delta P = K\frac{\rho v^2}{2} \tag{7.14}$$

where:

ΔP = *pressure drop*

ρ = *density*

v = *velocity*

Calculation of the fully turbulent friction factor f_T needed in the method requires knowledge of the relative roughness of the fitting. This is calculated from user entered values for roughness and fitting diameter. The Pipe Segment's standard friction factor equation (Churchill) is then called repeatedly with the calculated relative roughness at increasing Reynolds numbers until the limiting value of friction factor is found.

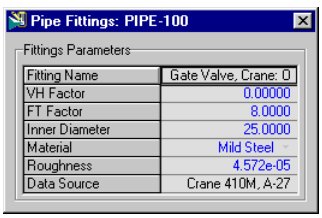
In general a fitting is characterised by either a velocity head factor (*A*) or a FT factor (*B*) but not both. HYSYS does not enforce this restriction however and you are free to define both factors for a fitting if required.

Pipe Fittings Property View

Refer to **Section 7.3.9 - Modifying the Fittings Database** for more information.

You can customize the pipe in the Pipe Fitting property view.

Figure 7.27



The above property view shows a standard fitting as it would be retrieved from the fittings database. If you customize a fitting by changing either the VH Factor or FT Factor, the word **User** is added to the fitting name to denote the fact that it is now user defined, and the Data Source field becomes modifiable to allow you to describe the source of the new data.

Default data for FT Factor and Data Source is provided for cases retrieved from earlier versions of HYSYS. Specifically the FT Factor is set to 0.0 and the Data Source is set to “HYSYS, pre V2.3”. The VH Factor is the same as the K Factor used in earlier versions.

Swage Fittings

A new capability has been added to the Pipe Segment to allow the pressure drop across reductions or enlargements in the pipe line to be calculated. The feature has been added as a new fitting type called a swage. The swage fitting automatically uses the upstream and downstream pipe/fitting diameters to calculate the K factor for the fitting. Once the K factor is known the pressure loss across the reducer/enlarger can be calculated. The equations used are as follows.

$$\Delta P = K_{out}\frac{\rho_{out}\upsilon_{out}^2}{2} - \frac{\rho_{in}\nu_{in}^2}{2} + \frac{\rho_{out}\nu_{out}^2}{2} \tag{7.15}$$

where:

ΔP = *static pressure loss*

ρ = *density*

ν = *velocity*

K = *reducer/enlarger K factor*

The *K* factor from the above equation is calculated from the following equations:

For reducers

$$K_{out} = 0.8\sin\frac{\theta}{2}(1-\beta^2) \qquad for(\theta \le 45°)$$

$$K_{out} = 0.5(1-\beta^2)\sqrt{\sin\frac{\theta}{2}} \qquad for(45° < \theta \le 180°) \qquad (7.16)$$

where:

$$\beta = \frac{d_{out}}{d_{in}}$$

For enlargers

$$K_{out} = \frac{2.6\sin\frac{\theta}{2}(1-\beta^2)^2}{\beta^4} \qquad for(\theta \le 45°)$$

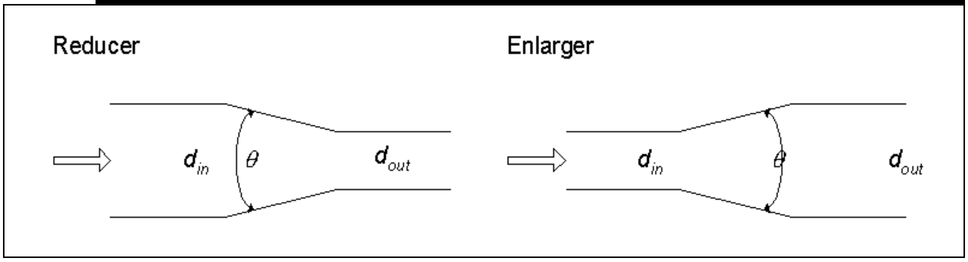
$$K_{out} = \frac{(1-\beta^2)^2}{\beta^4} \qquad for(45° < \theta \le 180°) \qquad (7.17)$$

where:

$$\beta = \frac{d_{in}}{d_{out}}$$

θ in the equations above is known as swage angle. Swage angle is shown in the figure below:

Figure 7.28



Equations for *K* above are taken from Crane, Flow of Fluids, Publication 410M, Appendix A-26.

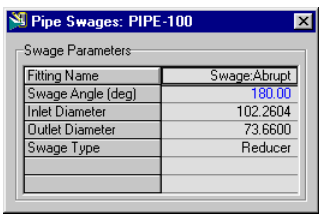
As stated above a swage segment automatically considers the upstream (d_{in}) and downstream (d_{out}) diameters to work out whether the swage is a reducer or an enlarger and calculate the appropriate *K* value. In addition the following special cases are detected and a fixed *K* value is used.

- The swage is the first segment in the pipe and an entrance *K* value of 0.5 is used.
- The swage is the last segment in the pipe and an exit *K* value of 1.0 is used.
- $d_{in} = d_{out}$ the swage is a simple coupling and a *K* value of 0.04 is used.

Pipe Swages Property View

A new swage fitting property view has been created to allow you to update the swage angle for a swage fitting. It also displays the upstream and downstream diameters that are used in the calculation as shown in the figure below.

Figure 7.29



The automatic detection of upstream and downstream diameters by the swage segment means that there cannot be two consecutive swage segments in a pipe. This restriction is enforced by HYSYS which prevents you from specifying two adjacent segments to be swages. In addition, if two adjacent swage segments would result from deletion of an intervening pipe or fitting segment, the second swage segment is automatically converted to a default Pipe Segment. An explanatory message appears in both cases.

Removing a Segment

To remove a segment from the Length-Elevation Profile group, select one of its parameters and click the Delete Segment button.

No confirmation is given by HYSYS before segments are removed.

You can remove all input from the Length-Elevation Profile group by clicking the Clear Profile button.

Heat Transfer Page

The Heat Transfer page is used to enter data for defining the heat transfer. The Specify By group, at the top of the property view, contains four radio buttons. Selecting one of the radio buttons displays one of the four ways of defining heat transfer:

- Specified heat loss
- Overall Heat Transfer Coefficient (HTC)
- HTC specified by segment
- Estimated HTC

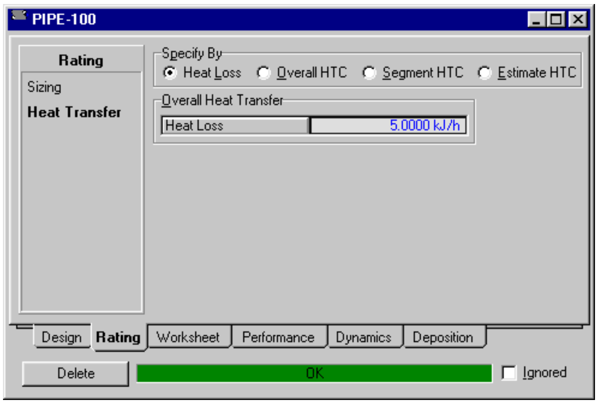
The radio button does not force the pipe segment to use that method of calculation – it only provides access to the property views.

HYSYS works out which method to use from the data provided.

Heat Loss

HYSYS selects the Heat Loss radio button as the default setting, when you select the Heat Transfer page for the first time. The property view appears as shown in the figure below:

Figure 7.30



If the Overall heat duty of the pipe is known, the energy balance can be calculated immediately. Each increment is assumed to have the same heat loss. You enter the heat loss for the pipe in the Heat Loss field. This assumption is valid when the temperature profile is flat, indicating low heat transfer rates compared to the heat flows of the streams. This is the fastest solution method.

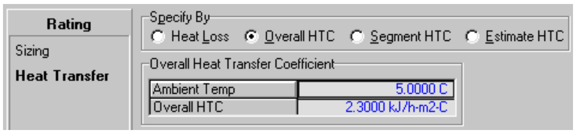
If both inlet and outlet temperatures are specified, a linear profile is assumed and HYSYS can calculate the overall heat duty. This method allows fast calculation when stream conditions are known. Select the Heat Loss radio button to see the calculated overall heat duty.

The value in the Heat Loss field is black in colour, signifying that the value was generated by HYSYS.

Overall HTC

When you select the Overall HTC radio button, the Heat Transfer page changes to the property view shown in the figure below.

Figure 7.31

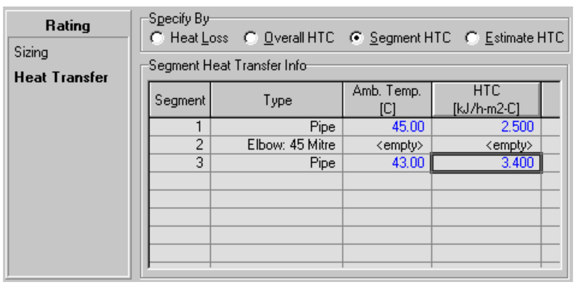


If the overall HTC (Heat Transfer Coefficient) and a representative ambient temperature are known, rigorous heat transfer calculations are performed on each increment.

Segment HTC

When you select the Segment HTC radio button, the Heat Transfer page changes to the property view shown in the figure below.

Figure 7.32

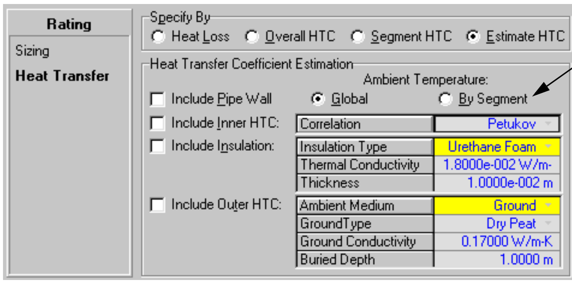


If the heat transfer coefficient and a representative ambient temperature are known for each segment. You can specify the ambient temperature and HTC for each pipe segment that was created on the Sizing page. HYSYS performs rigorous heat transfer calculations on each increment.

Estimate HTC

When you select the Estimate HTC radio button, the Heat Transfer page changes to the property view shown in the figure below.

Figure 7.33



You can select whether the ambient temperature used in the heat transfer calculations is for the entire pipe or for each segment of the pipe.

The Overall HTC and Estimate HTC can be used together to define the heat transfer information for the pipe.

If you only know the Ambient Temperature, you can supply it in the Overall HTC section and have the Overall HTC value calculated by the Estimate HTC section. Likewise, you need to specify the Ambient Temperature in the Estimate HTC section for the pipe segment to have enough heat transfer information to solve.

If the pipe's HTC is unknown, you can enter information in this property view and HYSYS calculates the HTC for the pipe.

Inside Film Convection

You can prompt HYSYS to estimate the inside film heat transfer coefficient using one of the five correlations provided.

The Petukov, Dittus, and Sieder methods for calculation of inner HTC are limited to single phase applications and essentially turbulent flow only. Two and three phase systems are modeled using the single phase equations with "averaged" fluid properties. A correction for laminar flow is applied but this is not particularly effective. It is recommended that these three methods be used only for single phase pipelines operating at high Reynolds numbers (> 10000).

The Profes and HTFS methods should provide much better results for two and three phase systems, and in the laminar flow region at the cost of some increase in calculation time. In general the Profes option is recommended for most pipeline applications since it takes into full account the flow regime in the pipe and is reasonably efficient in calculation. The HTFS option is more calculation intensive, particularly in two phase applications where additional flash calculations are required. It is recommended for use in cases with a high heat flux with high delta temperatures between the pipe contents and ambient conditions.

The five correlations provided are:

- **Petukov (1970)**

$$h = \frac{k}{d}\frac{(f/8)Re_d Pr}{1.07 + 12.7(f/8)^{1/2}(Pr^{2/3} - 1)} \tag{7.18}$$

- **Dittus and Boelter (1930)**

$$h = \frac{k}{d}0.023 Re_d^{0.8} Pr^n \tag{7.19}$$

where:

$$n = \begin{matrix} 0.4 \rightarrow \textit{for heating} \\ 0.3 \rightarrow \textit{for cooling} \end{matrix}$$

- **Sieder and Tate (1936)**

For two-phase flow:

$$h_{\text{2-phase}} = \frac{k}{d}0.027 Re_d^{0.8} Pr^{1/3}\left(\frac{\mu_b}{\mu_w}\right)^{0.14} \tag{7.20}$$

For single phase flow:

$$h_{1\text{-phase}} = [(h_{lam})^{12} + (h_{2\text{-phase}})^{12}]^{\frac{1}{12}}$$

where: (7.21)

$$h_{lam} = 3.66 + 0.0668\frac{d}{L} \times Re \times \frac{Pr}{1 + 0.04\left(\frac{d}{L}RePr\right)^{\frac{2}{3}}}$$

Refer to the **ProFES Reference Guide** for more information.

- **Profes**. Implements the methods used by the Profes Pipe Simulation program (formerly PLAC). The methods are based on the Profes flow maps for horizontal and vertical flow, and appropriate correlations are used to determine the HTC in each region of the flow map.
- **HTFS**. Implements the methods used by HTFS programs. Separate correlations are used for boiling and condensing heat transfer, and for horizontal and vertical flow. The methods used are documented in the HTFS Handbook[13].

You can choose to include the pipe's thermal resistance in your HTC calculations by selecting the **Include Pipe Wall** checkbox. Activating this option requires that the thermal conductivity be defined for the pipe material on the detail property view of each Pipe Segment. Default values of thermal conductivity are provided for the standard materials that can be selected in the Pipe Segment.

Outside Conduction/Convection

Outside convection to either Air, Water or Ground can be included by selecting the **Include Outer HTC** checkbox. For air and water, the velocity of the ambient medium is defaulted to 1 m/s and is user-modifiable. The outside convection heat transfer coefficient correlation is for flow past horizontal tubes (J.P. Holman, 1989):

$$h = \frac{k}{d}0.25Re^{0.6}Pr^{0.38} \qquad (7.22)$$

If Ground is selected as the ambient medium, the Ground type can then be selected. The thermal conductivity of this medium appears but is also modifiable by typing over the default value.

The Ground types and their corresponding conductivities are tabulated below:

Ground Type	Conductivity	Ground Type	Conductivity
Dry Peat	0.17 W/mK	**Frozen Clay**	2.50 W/mK
Wet Peat	0.54 W/mK	**Gravel**	1.10 W/mK
Icy Peat	1.89 W/mK	**Sandy Gravel**	2.50 W/mK
Dry Sand	0.50 W/mK	**Limestone**	1.30 W/mK
Moist Sand	0.95 W/mK	**Sandy Stone**	1.95 W/mK
Wet Sand	2.20 W/mK	**Ice**	2.20 W/mK
Dry Clay	0.48 W/mK	**Cold Ice**	2.66 W/mK
Moist Clay	0.75 W/mK	**Loose Snow**	0.15 W/mK
Wet Clay	1.40 W/mK	**Hard Snow**	0.80 W/mK

In HYSYS, the surrounding heat transfer coefficient value is based on the following heat transfer resistance equation:

$$H_{surroundings} = \frac{1}{R_{surroundings}} \tag{7.23}$$

$$R_{surroundings} = \frac{D_{ot}}{2k_s}\ln\left[\frac{2Z_b + \sqrt{4Z_b^2 - D_{ot}^2}}{D_{ot}}\right] \tag{7.24}$$

where:

$H_{surroundings}$ = surrounding heat transfer coefficient

$R_{surroundings}$ = surrounding heat transfer resistance

Z_b = depth of cover to centreline of pipe

k_s = thermal conductivity of pipe-surrounding material (Air, Water, Ground)

D_{ot} = outer diameter of pipe, including insulation

Conduction Through Insulation

Conduction through the insulation or any other pipe coating can also be specified. Several representative materials are provided, with their respective thermal conductivities. You must specify a thickness for this coating.

Insulation / Pipe	Conductivity	Insulation / Pipe	Conductivity
Evacuated Annulus	0.005 W/mK	**Asphalt**	0.700 W/mK
Urethane Foam	0.018 W/mK	**Concrete**	1.000 W/mK
Glass Block	0.080 W/mK	**Concrete Insulated**	0.500 W/mK
Fiberglass Block	0.035 W/mK	**Neoprene**	0.250 W/mK
Fiber Blanket	0.070 W/mK	**PVC Foam**	0.040 W/mK
Fiber Blanket-Vap Barr	0.030 W/mK	**PVC Block**	0.150 W/mK
Plastic Block	0.036 W/mK	**PolyStyrene Foam**	0.027 W/mK

7.3.4 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

The PF Specs page is relevant to dynamics cases only.

7.3.5 Performance Tab

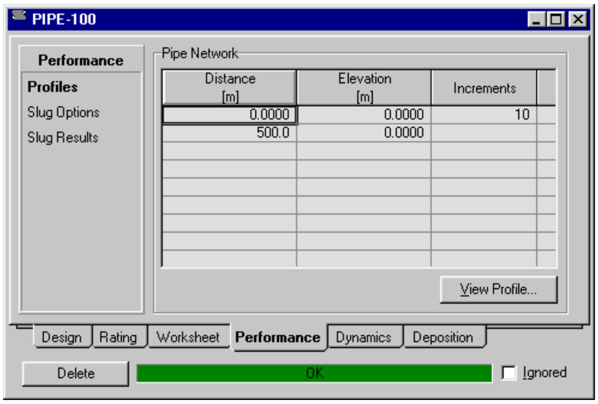
The Performance tab consists of the following pages:

- Profiles
- Slug Options
- Slug Results

Profiles Page

The Profiles page allows you to access information about the fluid stream conditions for each specified increment in the Pipe Segment.

Figure 7.34



The page contains a summary table for the segments which make up the Pipe Segment. The distance (length), elevation, and number of increments appear for each segment. You cannot modify the values on this page.

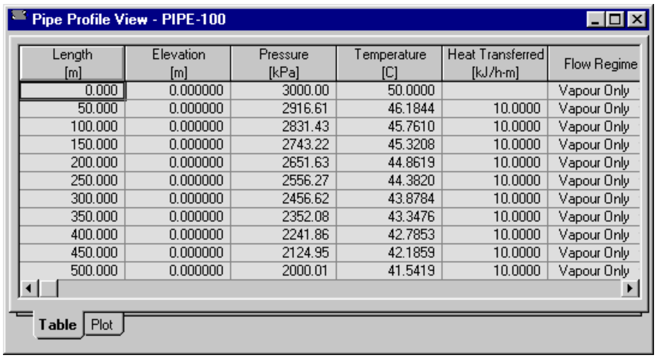
By clicking the **View Profile** button, the Pipe Profile property view appears, which consists of a Table tab and a Plot tab.

The Table tab displays the following information for each increment along the Pipe Segment:

- Length
- Elevation
- Pressure
- Temperature
- Heat Transferred
- Flow Regime
- Liquid Holdup
- Friction Gradient
- Static Gradient
- Accel Gradient

- Liquid Reynolds Number
- Vapour Reynolds Number
- Liquid Velocity
- Vapour Velocity
- Deposit Thickness
- Deposit Volume

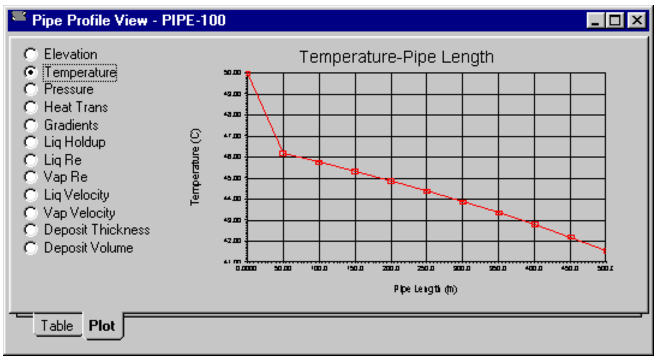
Figure 7.35



Length [m]	Elevation [m]	Pressure [kPa]	Temperature [C]	Heat Transferred [kJ/h-m]	Flow Regime
0.000	0.000000	3000.00	50.0000		Vapour Only
50.000	0.000000	2916.61	46.1844	10.0000	Vapour Only
100.000	0.000000	2831.43	45.7610	10.0000	Vapour Only
150.000	0.000000	2743.22	45.3208	10.0000	Vapour Only
200.000	0.000000	2651.63	44.8619	10.0000	Vapour Only
250.000	0.000000	2556.27	44.3820	10.0000	Vapour Only
300.000	0.000000	2456.62	43.8784	10.0000	Vapour Only
350.000	0.000000	2352.08	43.3476	10.0000	Vapour Only
400.000	0.000000	2241.86	42.7853	10.0000	Vapour Only
450.000	0.000000	2124.95	42.1859	10.0000	Vapour Only
500.000	0.000000	2000.01	41.5419	10.0000	Vapour Only

The Plot tab graphically displays the profile data that is listed on the Table page. Select one of the radio buttons to view a profile with Length as the x-axis variable:

Figure 7.36



Refer to **Section 1.3.1 - Graph Control Property View** for information regarding the customization of plots.

You can modify the plot by right-clicking on the plot area, and selecting **Graph Control** from the object inspect menu.

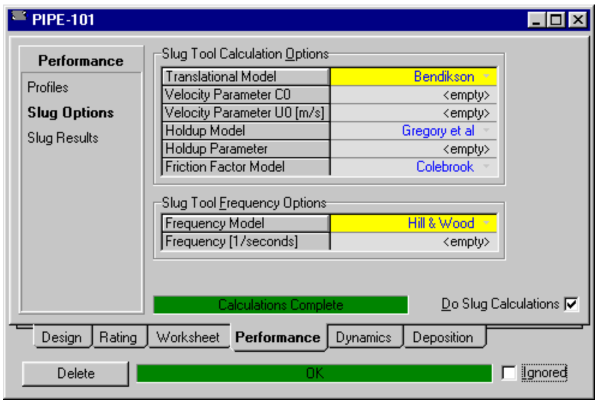
Slug Tool

The Slug Tool predicts slug properties for horizontal and inclined two-phase flows in each Pipe Segment. Travelling wave solutions of the one-dimensional averaged mass and momentum equations are found and analysed to obtain slug flow properties. Stratified flow is tested for instability to small disturbances and then analysed in the unstable region to find if slug flow is possible. If large amplitude waves can bridge the pipe then slug flow is deemed to be possible. In this slug flow region a range of frequencies is possible with a maximum slug frequency occurring for slugs of zero length. Up to this maximum there is a relationship between frequency and slug length with maximum lengths occurring for the lowest frequencies. The other slug properties such as bubble length, average film holdup, slug transitional velocity, average pressure gradient can all be found over the range of allowable slug frequencies.

The detailed methodology used to predict slug formation and slug properties was developed within AspenTech and is described in the paper “The modelling of slug flow properties” by M Watson[19].

Slug Options Page

Figure 7.37



The entries on the Slug Options page control the models and parameters used by the slug tool in its calculations as follows:

- **Translational Model**. You can select the option to be used for calculating the translational velocity of the slugs in the pipeline. The general form of the translational velocity is of the form:

$$c = C_0 V_M + U_0 \tag{7.25}$$

where:

c = translation velocity of slug

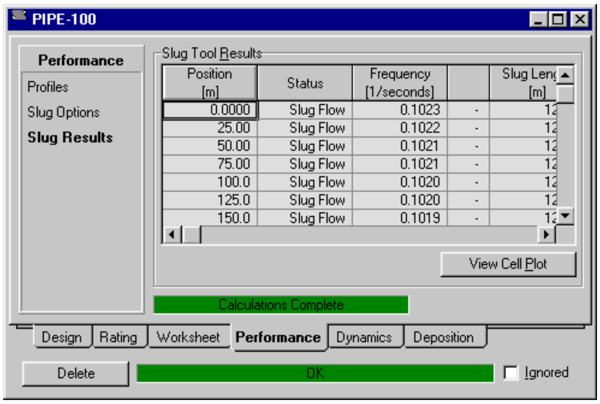
V_M = superficial velocity of two phase mixture

C_0, U_0 = constants

  You have the option to select the Bendikson (1984) model to predict values of C_0 and U_0 or to select User Specified to enter values manually.
- **Holdup Model**. You can select the option to be used to calculate the liquid holdup in the pipe. Two options are available: Gregory et. al. uses the methods published by Gregory et. al. (1978) or User Specified to enter a user defined value for the holdup fraction.
- **Friction Factor**. Two options are available to select the friction factor model to be used in the slug tool calculations: Smooth pipe or Colebrook equation.
- **Frequency Option**. The slug tool evaluates slug flow characteristics at a particular slug frequency. This frequency can either be predicted by the Hill & Wood correlation or specified by the user.

Slug Results Page

Figure 7.38



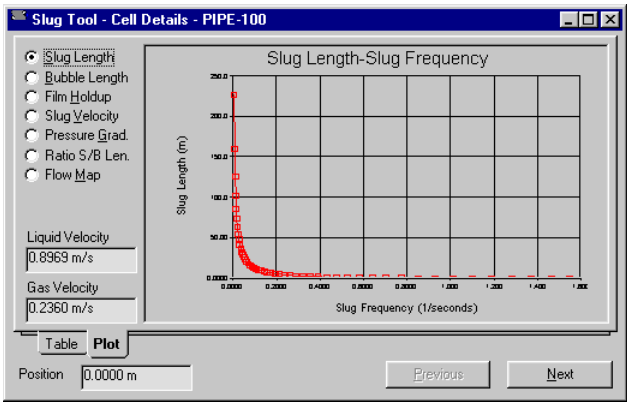
The Slug Results page presents the result from the slug tool analysis as a table with the following entries.

Column	Description
Position	Distance along the pipe.
Status	Result of the slug calculations. Possible results are Single Phase, Stable two phase, Slug flow, Annular flow, Bubble flow or Unknown. Any error in the calculation is also reported here.
Frequency	Slug frequency used to calculate the slug properties. This is normally the value calculated by the Hill & Wood correlation or the user specified slug frequency according to the settings on the Slug Options page. When the correlation or user specified frequency lies outside the predicted range of slug frequencies this field shows either the minimum or maximum slug frequency which is indicated by the entry in the next column.
Slug Length	Average length of a slug at the indicated frequency.
Bubble Length	Average length of a bubble at the indicated frequency.
Film Holdup	Film holdup as a fraction.
Velocity	Translational velocity of the slug.
Pressure Gradient	Pressure drop over the slug/bubble unit.
Slug/Bubble ratio	Ratio of lengths of slug and bubble.

Cell Details

When you click the **View Cell Plot** button on the Slug Results page, the Cell Details property view appears.

Figure 7.39



The property view shows the slug properties for a single position in the pipe across the full range of possible slug frequencies in both tabular and graphical form. A further graph shows the flow regime map for the cell indicating the region of possible slug formation at different vapour and liquid flowrates.

The Next button and Previous button on the property view allow you to move along the pipe to inspect the detailed results at any point.

7.3.6 Dynamics Tab

The Dynamics tab contains basic pipe parameter options to configure the pipe for Dynamics mode.

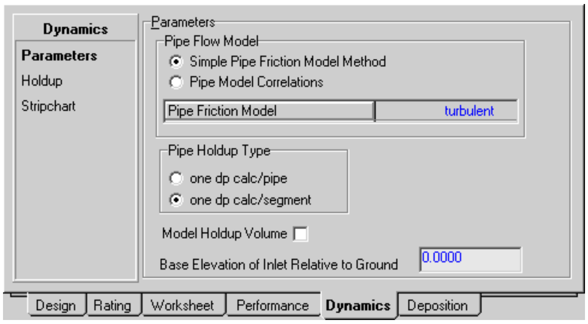
Pipe unit operation does not model choking or advanced effects such as shock waves, momentum balances, and so on.

To model choking and other advanced effects, use ProFES, OLGA pipe extensions, or Aspen Hydraulics flowsheet.

Parameters Page

The Parameters page allows you to specify the pipe flow model, holdup type, and base elevation for the Dynamics mode.

Figure 7.40



The following table lists and describes the objects available in the Parameters page:

Refer to **Friction Factor** section for more information.

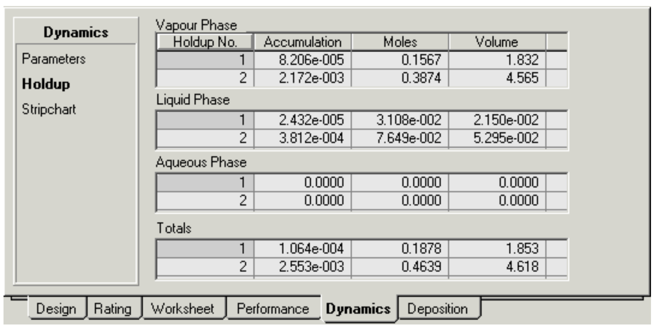
Object	Description
Simple Pipe Friction Model Method radio button	Allows you to select between turbulent and full range Churchill methods to simulate the pipe flow model in Dynamics mode. The **Pipe Friction Model** drop-down list is only available if you select the **Simple Pipe Friction Model Method** radio button.
Pipe Model Correlations radio button	Allows you to select the pipe flow model based on the available pipe flow correlation selection from the **Parameters Page** in the Design tab. The calculation time for this method is long and rigorous, however, the results are more accurate.
one holdup in pipe radio button	Allows you to calculate the overall holdup values of the entire pipe. This method calculates the results by lumping together all the volume. The calculation time is short, however, this method is not recommended if you want to track composition (or model lag) along the pipe.
one holdup per segment radio button	Allows you to calculate the holdup values for each segment in the entire pipe. This method calculates and models the composition and other changes through the pipe network rigorously. The calculation time is long, however, the results are more accurate.

Object	Description
Model Holdup Volume checkbox	Allows you to calculate pipe volumes based on the pipe lengths and diameter. Generally the pipe volumes are ignored or lumped together in one vessel for the calculation, unless a model of the composition lag is required. The lumped volume approach is a simpler more robust option, and often the pressure drop result is the main interest. Having to consider many holdups with small volumes may lead to instabilities.
Base Elevation of Inlet Relative to Ground field	Allows you to specify the elevation of the pipe relative to the ground for Dynamics mode.

Holdup Page

The Holdup page contains information regarding the properties, composition, and amount of the holdup.

Figure 7.41



For each phase contained within the volume space of the unit operation, the following is specified:

Holdup Details	Description
Holdup No.	Displays the designated number of each segment in the pipe. Number **1** indicates the first segment, number **2** indicates the second segment, and so forth.
Accumulation	The accumulation refers to the rate of change of material in the holdup for each phase.
Moles	The amount of material in the holdup for each phase.
Volume	The holdup volume of each phase.

Stripchart Page

Refer to **Section 1.3.7 - Stripchart Page/Tab** for more information.

The Stripchart page allows you to select and create default strip charts containing various variable associated to the operation.

7.3.7 Deposition Tab

Deposition is a general capability that can be used to model deposition of material that affects pressure drop or heat transfer to or from the pipe. Possible deposits include wax, asphaltenes, hydrates, sand, and so forth. The Deposition tab contains the following pages:

- Methods
- Properties
- Profile
- Limits

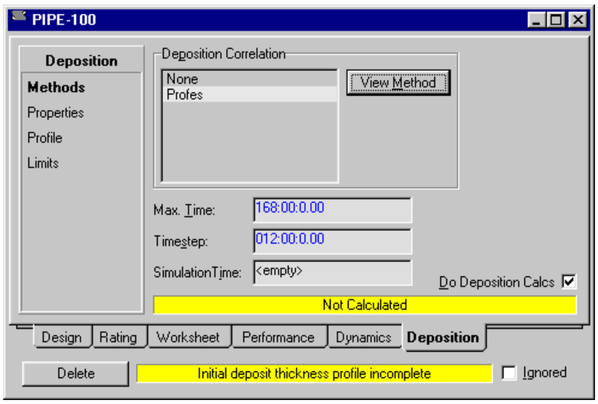
HYSYS provides a model for one type of deposit namely Wax deposition modeled using the Profes methods. Other third party methods can be added as plug-in extensions.

Methods Page

For more information about the Profes Wax method, refer to **Section 7.3.8 - Profes Wax Method**.

The Methods page displays the available deposition methods. Profes Wax is the only standard one at present.

Figure 7.42



Registered third party plug-in methods also appear on this page.

The Profes Wax model is installed by selecting it from the Deposition Correlation list. On installation the Pipe Segment is not able to solve until the initial deposition thickness is defined on the Profile page. Once the initial deposition thickness is defined the model solves using the default values provided for the other deposition data properties.

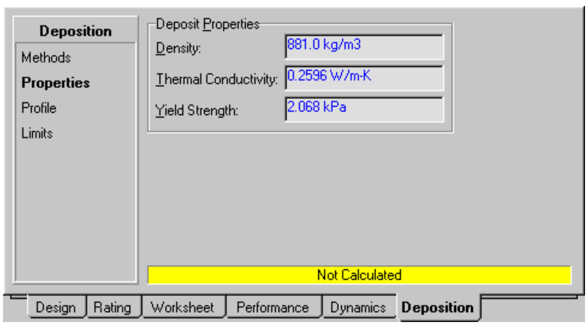
The Max. Time field allows you to specify the maximum amount of time wax deposits on the pipe. The Timestep field allows you to specify the timestep that the deposition rate is integrated over.

When solving with its default data, the model displays a warning message in the status bar of the Pipe Segment.

Properties Page

The Properties page allows you to specify deposit properties required by the deposition calculations.

Figure 7.43



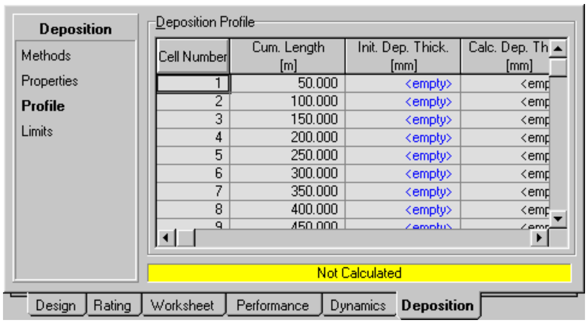
The page consists of three properties:

- Density of the deposit.
- Thermal Conductivity of the deposit.
- Yield Strength of the deposit.

Profile Page

The Profile page consists of the Deposition Profile table.

Figure 7.44

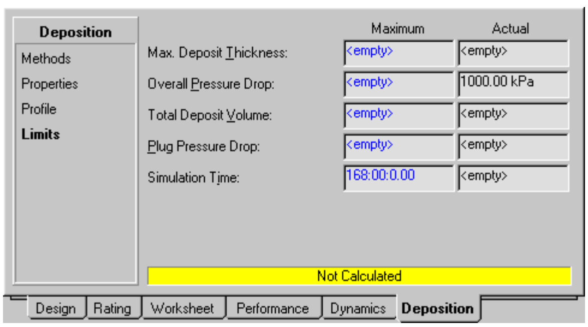


This table has two purposes:

- Is used to specify the initial deposition thickness, required by the deposition calculations.
- Displays the profile of the deposit on the pipe.

Limits Page

Figure 7.45



The Limits page allows you to specify the maximum limits for the following parameters.

- Max. Deposit Thickness
- Overall Pressure Drop
- Total Deposit Volume

- Plug Pressure Drop
- Simulation Time

7.3.8 Profes Wax Method

The deposition of the wax from the bulk oil onto the pipe wall is assumed to only be due to mass transfer, shear dispersion is not considered to be a significant factor. The rate of deposition is described by:

$$m' = k(C_{wall} - C_{bulk})AMw_{wax} \tag{7.26}$$

where:

m' = deposition rate (kg/s)

k = mass transfer coefficient (mole/m^2 s mole Fraction)

C = local concentration of wax forming components (mole fraction)

Mw_{wax} = molecular weight of wax (kg/mole)

A = cross-sectional area (m^2)

The mass transfer coefficient is calculated using the following correlation:

$$Sh = 0.015 \times Re^{0.88} Sc^{\frac{1}{3}} \tag{7.27}$$

where:

$$Sc = \frac{\mu_l}{\rho_l D}$$

$$Re = \frac{V_l \rho_l D_H}{\mu_l}$$

$$Sh = \frac{kD_H}{cD}$$

D = diffusivity of wax in oil (m^2/s)

μ = *liquid viscosity (kg/ms)*

ρ_l = *liquid density (kg/m²)*

k = *mass transfer coefficient (mole/m² s mole fraction)*

D_H = *hydraulic radius (m)*

V_l = *liquid velocity (m/s)*

c = *liquid molar density (mole/m³)*

The Reynolds number that is used in these calculations is based on the local liquid velocity and liquid hydraulic radius. Physical properties are taken as the single phase liquid values. The viscosity used is based on the fluid temperature and shear rate at the wall.

The difference in concentration of wax forming species between the bulk fluid and the wall, which is the driving force for the deposition of wax is obtained from calculating the equilibrium wax quantities at the two relevant temperatures.

These calculations provide a wax deposition rate which is integrated over each timestep to give the total quantity of wax laid down on the pipe wall.

Profes Wax Property View

When you click the **View Method** button, the Profes Wax property view appears. You can change the default data in the Profes Wax model, and tune it to your specific application in this property view. The Profes Wax property view consists of three tabs:

- Wax Data
- Tuning Data
- Ref. Comp

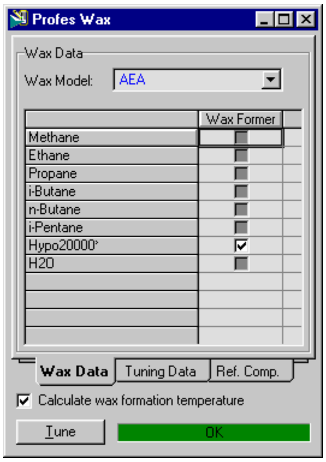
The **Calculate wax formation temperature** checkbox allows you to select whether the deposition model is to calculate the initial wax formation temperature or cloud point for each pipe element when performing the deposition calculations. If activated the results appear in the **Profile** page of the **Deposition** tab.

The Tune button initiates the tuning calculations and is only active when there is sufficient data to allow tuning calculation to take place. In other words, cloud point is defined, at least one temperature or wax mass percent pair is defined and reference composition defined.

Wax Data Tab

The Wax Data tab allows you to select the wax model to be used for the wax equilibrium calculations.

Figure 7.46



The Wax Model drop-down list provides you with four thermodynamic models for wax formation:

- Chung
- Pederson
- Conoco
- AEA (default)

All models are based on the following equation for the equilibrium constant, K_i, which is the ratio of concentrations of a particular component in the solid and liquid phase:

$$K_i = \frac{x_i^S}{x_i^L} = \frac{\zeta_i^L f_i^L}{\zeta_i^S f_i^S} \exp\left(\int_0^P \frac{V_i^L - V_i^S}{RT}\right) \partial P \tag{7.28}$$

where:

x_i = *mol fraction*

ζ_i = *activity coefficient*

f = standard state fugacity

P = pressure

V = molar volume

T = temperature

R = gas constant

S, L = denote solid and liquid phases

Once the equilibrium constant for each component has been calculated, they are used to determine the quantities and composition of each phase. The differences between the various thermodynamic models depend on how the terms in the equilibrium constant equation are evaluated. The four models available in the Profes method are described by the following equations:

- **AEA**:

$$\ln K_i = \frac{\Delta h_i^f}{RT}\left(1 - \frac{T}{T_i^f}\right) + \frac{\Delta C_p}{R}\left[1 - \frac{T_i^f}{T} + \ln\frac{T_i^f}{T}\right] + \int_0^P \frac{V_i^L - V_i}{RT} \partial P \tag{7.29}$$

- **Chung**:

$$\ln K_i = \frac{\Delta h_i^f}{RT}\left(1 - \frac{T}{T_i^f}\right) + \frac{V_i^L}{RT}(\delta_m^L - \delta_i^L)^2 + \ln\frac{V_i^L}{V_m} + 1 - \frac{V_i^L}{V_m} \tag{7.30}$$

- **Conoco (Erikson)**:

$$\ln K_i = \frac{\Delta h_i^f}{RT}\left(1 - \frac{T}{T_i^f}\right) \tag{7.31}$$

- **Pederson**:

$$\ln K_i = \frac{V_i^L(\delta_m^L - \delta_i^L)^2}{V_i^S(\delta_m^S - \delta_i^S)^2} + \frac{\Delta h_i^f}{RT}\left(1 - \frac{T}{T_i^f}\right) + \frac{\Delta C_p}{R}\left[1 - \frac{T_i^f}{T} + \ln\frac{T_i^f}{T}\right] \tag{7.32}$$

where:

Δh_i^f = *enthalpy of melting*

T_i^f = *melting temperature*

V = *molar volume*

δ = *solubility parameter*

ΔC_p = *heat capacity difference between solid and liquid*

m = *denotes mixture properties*

i = *component*

All the models require a detailed compositional analysis of the fluid in order to be used effectively and for the Conoco model Erickson et al proposed that the hydrocarbon analysis should distinguish between normal and non normal paraffin components as there is a substantial difference in melting points between these two groups. The melting temperatures make a very significant impact on the predicted cloud point for any given composition. In Pederson model the K_i values depend on the composition of the liquid and solid phases; this is unlike normal equilibrium calculations, where the K_i 's are fixed for any temperature and pressure, and can lead to unstable or incorrect numerical solutions.

The AEA model is the only model which incorporates a term for the effect of pressure on the liquid-solid equilibrium, the result of this is to counteract the increased solubility of wax forming components at high pressures which is due to more light ends entering the liquid phase. Using this model, the predicted cloud point and wax quantities can both increase or decrease with increasing pressure, depending on the fluid composition.

The table allows you to select which components in the system are able to form wax. The default criteria for the components in this table are as follows:

- Components with a mole weight less than 140 or inorganic component types can never form wax. The checkbox for these components is set to a grey checkbox that cannot be modified.
- Hydrocarbon component types form wax. The checkbox is automatically selected for these components, but you can clear the checkbox.

Hypothetical components generally fall into this category.

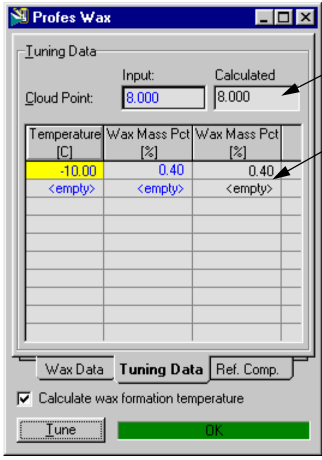
- Other organic component types do not form wax. The checkbox is clear but you can select it.

The ability to select whether a particular component forms wax gives you additional control when defining the wax formation characteristics of a system. For example, you can define two hypothetical components with common properties, and by setting one as a wax former you could vary the quantity of wax produced in the boiling range covered by the hypotheticals by varying their proportions.

Tuning Data Tab

The Tuning Data tab allows you to define the observed wax formation characteristics of a system to tune the wax model.

Figure 7.47



Calculated values for the Cloud Point and Wax Mass Percent appear in the calculated field and column after the tuning process is run.

The Cloud Point Input field allows you to specify the temperature at which the first wax appears. In other words, the phase transition temperature between single liquid phase and the two phase wax/liquid mixture.

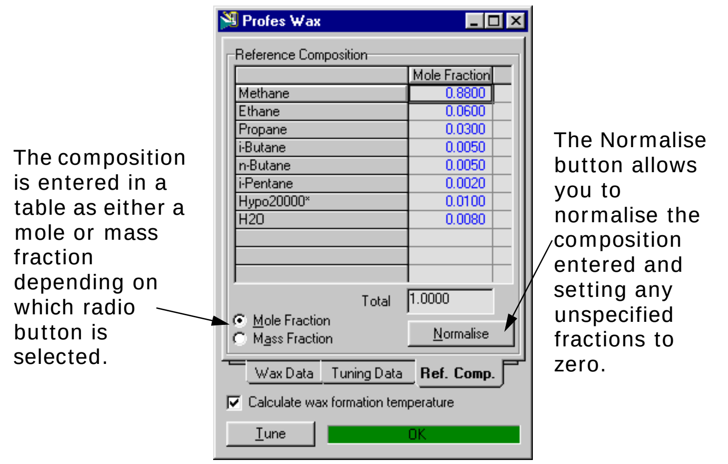
The table of Temperature vs. Wax Mass Percent allows you to define the quantity of wax deposit observed as a function of temperature. New points can be added to the table in any order; they are sorted by temperature when the tuning process is run. To run the tuning calculations a minimum of one pair of data points is required. Up to 10 pairs of data points can be specified.

To remove a point from the table both the temperature and the wax mass percent values must be deleted.

Ref Comp Tab

The Ref Comp tab allows you to specify the reference composition of the fluid that is used for tuning calculations.

Figure 7.48



Tuning Process

The tuning process is a series of calculations that is initiated as a task by clicking the Tune button. The first step validates the tuning data specified as follows:

- that there is at least one component identified as a wax former.
- a valid reference composition has been entered.
- sorting the pairs of temperature/wax mass percent in order of descending temperature.
- ensure cloud point temperature is greater than any temperature in table of temperature/wax mass percent pairs.

If any problem is found the tuning process stops, and an appropriate error message appears on the tuning status bar.

If the tuning data is valid then the tuning process first does a VLE flash at 15°C and 100 kPa to calculate the liquid composition of the reference stream. This is used as the base composition for all subsequent tuning calculations. If a liquid phase cannot be found in the reference stream at these conditions the tuning calculations fail.

The tuning process then continues using an iterative least squares solution method. Progress of the tuning process is output to the main HYSYS status bar. When complete the tuning process checks for convergence and displays the result on the tuning status bar. If the tuning process converged then the calculated results on the Tuning data tab are updated. If the tuning process failed to converge the tuning parameters are set to the best values that were obtained. Tuning parameters can be reset to their original values by re-selecting the wax model.

There are three tuning parameters available for the Chung, Pederson, and Conoco wax models, and four tuning parameters for the AEA model. The tuning process only attempts to tune as many parameters as is possible from the specified data (for example, cloud point + one pair of temperature/wax percent data allows tuning of two parameters, more pairs of temperature/wax percent data points are required to tune additional parameters). In cases where an attempt to tune three or four parameters fails to converge, a second tuning attempt is made automatically for just two tuning parameters.

The convergence of the tuning process is checked by looking at the cloud point result since this is the most critical parameter. Generally convergence is achieved when there are one or two pairs of temperature/wax percent data. Given the emphasis placed on achieving the cloud point the calculated results for wax percent can often show greater errors, particularly when there are multiple temperature/wax percent points.

You should realise that the degree to which the tuning parameters can adjust the temperature/wax percent curve predicted by the models is limited, and that hand tuning by changing the number and proportion of wax forming components in the system may be required in some cases.

7.3.9 Modifying the Fittings Database

The fittings data base contains VH Factor and FT Factor data taken from Perry. Some of the Factor data are taken from Crane.

The following table details the values in the database. The Data Source column displays the source of the data values.

Description	VH Factor	FT Factor	Swage Angle	Data Source
Pipe	0	0	0	
Swage: Abrupt	0	0	180	
Swage: 45 degree	0	0	45	
Elbow: 45 Std	0	16	0	Crane 410M, A-29
Elbow: 45 Long	0.2	0	0	Perry 5th ed, Table 5-19
Elbow: 90 Std	0	30	0	Crane 410M, A-29
Elbow: 90 Long	0.45	0	0	Perry 5th ed, Table 5-19
Bend: 90, r/d 1	0	20	0	Crane 410M, A-29
Bend: 90, r/d 1.5	0	14	0	Crane 410M, A-29
Bend: 90, r/d 2	0	12	0	Crane 410M, A-29
Bend: 90, r/d 3	0	12	0	Crane 410M, A-29
Bend: 90, r/d 4	0	14	0	Crane 410M, A-29
Bend: 90, r/d 6	0	17	0	Crane 410M, A-29
Bend: 90, r/d 8	0	24	0	Crane 410M, A-29
Bend: 90, r/d 10	0	30	0	Crane 410M, A-29
Bend: 90, r/d 12	0	34	0	Crane 410M, A-29
Bend: 90, r/d 14	0	38	0	Crane 410M, A-29
Bend: 90, r/d 16	0	42	0	Crane 410M, A-29
Bend: 90, r/d 20	0	50	0	Crane 410M, A-29
Elbow: 45 Mitre	0	60	0	Crane 410M, A-29
Elbow: 90 Mitre	0	60	0	Crane 410M, A-29
180 Degree Close Return	0	50	0	Crane 410M, A-29
Tee: Branch Blanked	0	20	0	Crane 410M, A-29
Tee: As Elbow	0	60	0	Crane 410M, A-29
Coupling/Union	0.04	0	0	Perry 5th ed, Table 5-19
Gate Valve: Open	0.17	0	0	Perry 5th ed, Table 5-19
Gate Valve: Three Quarter	0.9	0	0	Perry 5th ed, Table 5-19
Gate Valve: Half	4.5	0	0	Perry 5th ed, Table 5-19
Gate Valve: One Quarter	24	0	0	Perry 5th ed, Table 5-19

Description	VH Factor	FT Factor	Swage Angle	Data Source
Gate Valve, Crane: Open	0	8	0	Crane 410M, A-27
Diaphram Valve: Open	2.3		0	Perry 5th ed, Table 5-19
Diaphram Valve: Three Quarter	2.6	0	0	Perry 5th ed, Table 5-19
Diaphram Valve: Half	4.3	0	0	Perry 5th ed, Table 5-19
Diaphram Valve: One Quarter	21	0	0	Perry 5th ed, Table 5-19
Globe Valve: Open	6	0	0	Perry 5th ed, Table 5-19
Globe Valve: Half	9.5	0	0	Perry 5th ed, Table 5-19
Globe Valve, Crane: Open	0	340	0	Crane 410M, A-27
Angle Valve: Open	2	0	0	Perry 5th ed, Table 5-19
Angle Valve, 45 deg: Open	0	55	0	Perry 5th ed, Table 5-19
Angle Valve, 90 deg: Open	0	150	0	Crane 410M, A-27
Blowoff Valve: Open	3	0	0	Perry 5th ed, Table 5-19
Plug Cock: Angle 5	0.05	0	0	Perry 5th ed, Table 5-19
Plug Cock: Angle 10	0.29	0	0	Perry 5th ed, Table 5-19
Plug Cock: Angle 20	1.56	0	0	Perry 5th ed, Table 5-19
Plug Cock: Angle 40	17.3	0	0	Perry 5th ed, Table 5-19
Plug Cock: Angle 60	206	0	0	Perry 5th ed, Table 5-19
Plug Cock: Open	0	18	0	Crane 410M, A-29
Butterfly Valve: Angle 5	0.24	0	0	Perry 5th ed, Table 5-19
Butterfly Valve: Angle 10	0.52	0	0	Perry 5th ed, Table 5-19
Butterfly Valve: Angle 20	1.54	0	0	Perry 5th ed, Table 5-19
Butterfly Valve: Angle 40	10.8	0	0	Perry 5th ed, Table 5-19
Butterfly Valve: Angle 60	118	0	0	Perry 5th ed, Table 5-19
Butterfly Valve: 2-8in, Open	0	45	0	Crane 410M, A-28
Butterfly Valve: 10-14in, Open	0	35	0	Crane 410M, A-28
Butterfly Valve: 16-24in, Open	0	25	0	Crane 410M, A-28
Ball Valve: Open	0	3	0	Crane 410M, A-28
Check Valve: Swing	2	0	0	Perry 5th ed, Table 5-19
Check Valve: Disk	10	0	0	Perry 5th ed, Table 5-19
Check Valve: Ball	70	0	0	Perry 5th ed, Table 5-19
Check Valve: Lift	0	600	0	Crane 410M, A-27
Check Valve: 45 deg Lift	0	55	0	Crane 410M, A-27
Foot Valve	15	0	0	Perry 5th ed, Table 5-19
Foot Valve: Poppet disk	0	420	0	Crane 410M, A-28
Foot Valve: Hinged disk	0	75	0	Crane 410M, A-28
Water Meter: Disk	7	0	0	Perry 5th ed, Table 5-19

Description	VH Factor	FT Factor	Swage Angle	Data Source
Water Meter: Piston	15	0	0	Perry 5th ed, Table 5-19
Water Meter: Rotary	10	0	0	Perry 5th ed, Table 5-19
Water Meter: Turbine	6	0	0	Perry 5th ed, Table 5-19
User Defined	0	0	0	User specified

A few sections from the “fittings.db” file are shown below:

```
FittingType elbow45std
   VHFactor 0.0
   Desc “Elbow: 45 Std”
   FTFactor 16.0
   Data Source “Crane 410M, A-29”
end
FittingType swage2
   VHFactor 0.0
   Desc “Wage: 45 degree”
   Sweating 45.0
end
...
FittingTypeGroup FTG
    AddFitt elbow45std
...
end
```

This can be broken down, line by line:

1. FittingType elbow45std

   What this does is define an object “elbow45std” of type “FittingType”. “FittingType” has three members (parameters): a VH Factor (K-Factor), FT factor or Swage angle, and a description.

The object name “elbow45std” is only an internal name; it doesn't appear in any lists or property views.

2. VHFactor 0.0

   This is the K-Factor for the fitting. When you add a fitting to the fittings list, this is the number that is put in the K-Factor column.

3. Desc “Elbow: 45 Std”

This assigns a label (description) to the fitting “elbow45std”. It is this label that is used in the fittings window to select fittings.

4. This line contains one of the following two possible command lines:
    - FTFactor 16.0
      This is the FT factor for the fitting. When you add a fitting to the fittings list, this is the number that is put in the FT factor column.
    - Swage Angle 45
      This command is used to assign the value for the swage angle fitting calculation method.
5. DataSource “Crane 410M, A-29”

   This tells you where the data source was taken from.
6. end

   This tells HYSYS that the description of “elbow45std” is done.

So, you have a definition of a fitting. But, that's not quite enough. All the fittings are gathered into one group - a “FittingTypeGroup”- to make it easier for HYSYS to determine what should go where.

1. FittingTypeGroup FTG

   Same as line 1 above. This defines an object “FTG” of type “FittingTypeGroup”. “FTG” can have many parameters, but they must all be of the same type - FittingType. The FittingTypeGroup is like a container for all the pipe fittings.
2. AddFitt elbow45std

   This adds the previously defined fitting to the group. Notice that the fitting MUST be defined before it is added to the group. All new fittings should be added last in the database file. When the fittings appear in the drop-down list, they are sorted alphabetically by their Desc parameter.
3. end

   This tells HYSYS that you have added all the fittings you want to the fitting group. Notice that HYSYS does not automatically put fittings in the group just because they are defined beforehand.

New fittings should be added as the last entry in the database.

For example, if we had defined “elbow45std” as above, but forgot to add it to the fittings group, there would be no way to access it in the fittings window.

Also, the “end” command is very important. If you forget to put an “end” in somewhere in the middle of the fittings.db file, you can get errors that may or may not tell you what is actually wrong.

Adding a Fitting

So, now you can add your own fitting. Open the “fitting.db” file in an ASCII editor and move somewhere in the middle of the file (but make sure that you are above the definition of the fittings group).

Now, add the following lines:

```
FittingType loopdeloop
   VHFactor 10.0
   Desc “Loop-de-loop!”
   FT Factor 0.0
   DataSource “Add fitting demo”
end
```

You have now created a fitting. You don't have to indent the VHFactor, Desc, FTFactor, and DataSource lines, it just makes for neater and easier to read files. Next, the fitting needs to be added to the fittings group.

Find the line in the file that says “FittingTypeGroup FTG”. Now, go anywhere between this line and the “end” line and type the following:

```
AddFitt loopdeloop
```

Now all you have to do is run HYSYS and make a Pipe Segment. The new fitting “Loop-de-loop!” appears in the fittings drop-down list, and if you add a “Loop-de-loop!” to the fittings list, it comes up with a K-Factor of 10.0.

To take out the fitting, just delete the lines that were previously added.

7.4 Relief Valve

The Relief Valve unit operation can be used to model several types of spring loaded Relief Valves. Relief Valves are used quite frequently in many different industries in order to prevent dangerous situations occurring from pressure buildups in a system. Its purpose is to avert situations that occur in a dynamic environment. The flow through the Relief Valve can be vapour, liquid, liquid with precipitate or any combination of the three.

7.4.1 Relief Valve Property View

There are two ways that you can add a Relief Valve to your simulation:

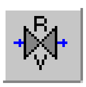
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Piping Equipment** radio button.
3. From the list of available unit operations, select **Relief Valve**.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

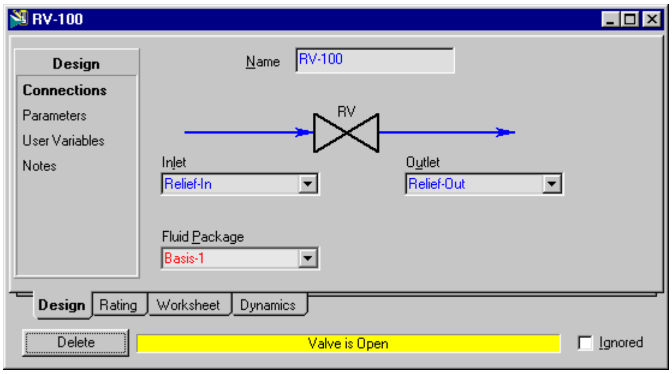
   You can also open the Object Palette by pressing the **F4**.
2. Double-click the **Relief Valve** icon.



Relief Valve icon

The Relief Valve property view appears.

Figure 7.49



7.4.2 Design Tab

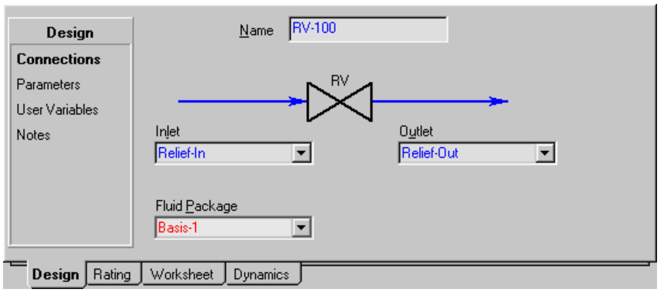
The Design tab of the Relief Valve property view contains the following pages:

- Connections
- Parameters
- User Variables
- Notes

Connections Page

The Connections page is where the inlet and outlet streams of the Relief Valve are specified.

Figure 7.50



The page contains the following fields described in the table below:

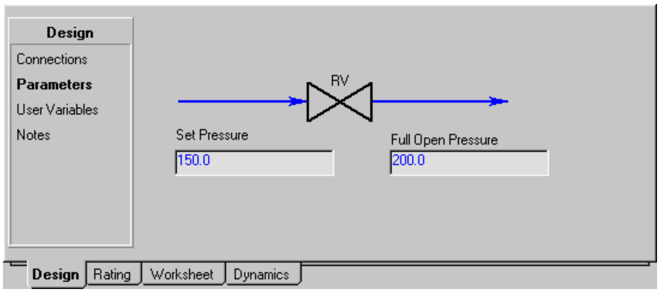
Field	Description
Name	The name of the Relief Valve. HYSYS provides a default designation for the unit operation, however, you can edit this name at any time by entering a new name in this field.
Inlet	Stream entering Relief Valve. You can either select a pre-existing stream from the drop-down list associated with this field or you can create a new stream by selecting this field and typing the stream name.
Outlet	Relief Valve exit stream. Like the Inlet field, you can either select a pre-existing stream from the drop-down list associated with this field or you can create a new stream by selecting this field and typing the stream name.
Fluid Package	Displays the fluid package associated to the relief valve. If the simulation case contains multiple fluid packages, you can open the drop-down list and select a different fluid package.

Parameters Page

The Parameters page contains only two fields, which are described in the table below:

Object	Description
Set Pressure	The pressure that the Relief Valve begins to open.
Full Open Pressure	The pressure that the Relief Valve is fully open.

Figure 7.51



User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

7.4.3 Rating tab

The Rating tab contains the following pages:

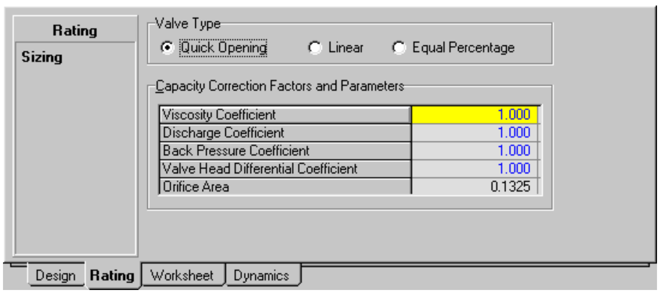
- Sizing
- Nozzles

If you are working exclusively in Steady State mode, you are not required to change any information on the pages accessible through the Rating tab.

Sizing Page

On the Sizing page, you can specify the Valve Type, and the Capacity Correction Factors and Parameters.

Figure 7.52



Valve Type

In HYSYS, you can specify three different valve characteristics for any Relief Valve in the simulation case.

Valve Type	Description
Quick Opening	A Relief Valve with quick opening valve characteristics obtains larger flows initially at lower valve openings. As the valve opens further, the flow increases at a smaller rate.

Valve Type	Description
Linear	A Relief Valve with linear valve characteristics has a flow, which is directly proportional to the valve % opening.
Equal Percentage	A Relief Valve with equal percentage valve characteristics initially obtains very small flows at lower valve openings. However, the flow increases rapidly as the valve opens to its full position.

Capacity Correction Factors and Parameters

The Capacity Correction Factors and Parameters group consists of five parameters of the flow equations. You can set:

- Viscosity Coefficient (K_V)
- Discharge Coefficient (K_D)
- Back Pressure Coefficient (K_B)
- Valve Head Differential Coefficient
- Orifice Area (A)

For more information on the function of these parameters, consult the following section on flow through the Relief Valve.

Flow Through Relief Valve

The mass flowrate through the Relief Valve varies depending on the vapour fraction and the pressure ratio across the valve. For two phase flow, the flows are proportional to the vapour fraction and can be calculated separately and then combined for the total flow.

Vapour Flow In Valve

For gases and vapours, flow may be choked or non-choked. If the pressure ratio is greater than the critical, the flow is **NOT** choked:

$$\frac{P_2}{P_1} \geq \left[\frac{2}{K+1}\right]^{\frac{K}{K-1}} \tag{7.33}$$

where:

P_1 = upstream pressure

P_2 = downstream pressure

K = ratio of Specific Heats

For Choked vapour flow, the mass flowrate is given by the following relationship:

$$W = AK_LK_DK_B\left[\frac{P_1K}{V_1}\left[\frac{2}{K+1}\right]^{\frac{K+1}{K-1}}\right]^{\frac{1}{2}} \tag{7.34}$$

where:

W = mass flow rate

A = relief valve orifice area

K_L = capacity correction factor for valve lift

K_D = coefficient of discharge

K_B = back pressure coefficient

V_1 = specific volume of the upstream fluid

For non-Choked vapour flow, the mass flowrate is given by:

$$W = AK_LK_D\left(\frac{P_1}{V_1}\left(\frac{2K}{K-1}\right)\left[\left(\frac{P_2}{P_1}\right)^{\frac{2}{K}} - \left(\frac{P_2}{P_1}\right)^{\frac{K+1}{K}}\right]\right)^{\frac{1}{2}} \tag{7.35}$$

Liquid Flow In Valve

Liquid Flow through the valve is calculated using the following equation:

$$W = AK_LK_DK_V[2(P_1 - P_2)\rho_1]^{\frac{1}{2}} \tag{7.36}$$

where:

ρ_1 = d*ensity of upstream fluid*

K_V = *viscosity correction factor*

Capacity Correction Factor (K_L)

The Capacity Correction Factor for back pressure is typically linear with increasing back pressure. The correct value of the factor should be user-specified. It may be obtained from the valve manufacturer. The capacity correction factor for valve lift compensates for the conditions when the Relief Valve is not completely open.

Increasing-sensitivity valves have the following flow characteristics:

$$K_L = \frac{L^2}{[a + (1 - a)L^4]^{1/2}} \tag{7.37}$$

Linear and decreasing-sensitivity valves have the following flow characteristics:

$$K_L = \frac{L}{[a + (1 - a)L^2]^{1/2}} \tag{7.38}$$

where:

$$a = \frac{\text{valve head differential a maximum flow}}{\text{valve head differential at zero flow}} \tag{7.39}$$

The valve head differential term allows for customization of the flow characteristics with respect to stem travel. Its value can range between 0 and 1.

Nozzles Page

Refer to **Section 1.3.6 - Nozzles Page** for more information.

The Nozzles page contains information regarding the elevation and diameter of the nozzles.

7.4.4 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

The PF Specs page is relevant to dynamics cases only.

7.4.5 Dynamics Tab

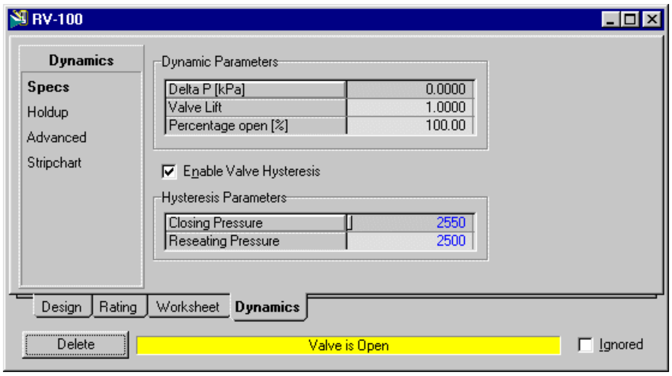
If you are working exclusively in Steady State mode, you are not required to change any information on the pages accessible through the Dynamics tab.

The Dynamics tab contains the following pages:

- Specs
- Holdup
- Advanced
- Stripchart

Specs Page

Figure 7.53



The Specs page consists of two groups:

- Dynamic Parameters
  This group consists of three parameters.

Parameter	Descriptions
Delta P	Pressure drop across the valve.
Valve Lift	The Relief Valve lift. It is calculated using one of the two following formulas: If inlet pressure is increasing: $$L = \left[\frac{P_1 - P_{OPEN}}{P_{FULL} - P_{OPEN}}\right] \quad (7.40)$$ *where:* P_1 = *upstream pressure* P_{OPEN} = *set pressure* P_{FULL} = *full open pressure* If inlet pressure is decreasing: $$L = \left[\frac{P_1 - P_{RESEAT}}{P_{CLOSE} - P_{OPEN}}\right] \quad (7.41)$$ *where:* P_1 = *upstream pressure* P_{RESEAT} = *reseating pressure* P_{CLOSE} = *closing pressure*
Percentage Open	The Valve Lift in percentage.

- Hysteresis Parameters
  When the **Enable Valve Hysteresis** checkbox is selected, the Hysteresis Parameters group appears. This group contains two fields, which are described in the table below:

Field	Descriptions
Closing Pressure	Pressure that the valve begins to close after reaching the full lift pressure. In other words, the value entered in the full pressure field on the Parameters page of the Design tab.
Reseating Pressure	The pressure that the valve *reseats* after discharge.

Holdup Page

For more information, refer to the valve operation **Holdup Page** in the **Dynamics** tab.

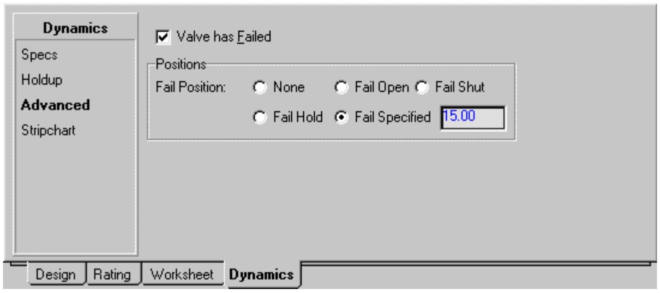
The Holdup page contains information regarding the holdup properties, composition, and amount.

Each unit operation in HYSYS has the capacity to store material and energy. Typical Valves usually have significantly less holdup than other unit operations in a plant. Therefore, the volume of the Valve operation in HYSYS is defaulted to be zero.

Advanced Page

The fail-safe function in Relief Valve is used to prevent pipeline and equipment from physical damages due to escalation in pressure. The plant operator can either apply this feature to relief the pressure built-up from affecting other parts of the plant or it can be used in their training to simulate valve stickiness or failure.

Figure 7.54



The Relief Valve has five fail modes. The way that these fail modes interact with the relief valve is somewhat different from the ones of the control valve discussed in **Chapter 1.6.3 - Control Valve Actuator** of the **HYSYS Dynamic Modeling** guide.

Refer to **Chapter 1.6 - HYSYS Dynamics** in the **HYSYS Dynamic Modeling** guide for more information.

To activate the relief valve fail mode option, ensure that the Integrator is running with HYSYS Dynamics license.

To set the Relief Valve in fail state, you can select the **Valve has Failed** checkbox on the **Advanced** page. You can now specify one of the following fail modes:

- **None**. Relief valve operates as it is designed to be (same as the operating condition when the Valve has Failed checkbox is not active).
- **Fail Open**. The valve lift completely opens. The valve lift remains at maximum opening position even when the inlet pressure is not longer above the opening (set) pressure. The Relief Valve continues to fully open until it is reset.
- **Fail Shut**. The valve lift completely closes. The valve life stays shut even when the inlet pressure is above the opening (set) pressure. The Relief Valve remains fully shut until it is reset.
- **Fail Hold**. Allows you to simulate the valve lift stickiness by holding the valve lift to the last failed position. The Relief Valve lift will not move even when the inlet pressure is no longer above the opening pressure.
- **Fail Specified**. Allows you to manually specify the fail position when the Relief Valve has failed. The fail position is expressed in terms of percentage of the valve opening, and it is used to define the amount of valve lift.

Stripchart Page

Refer to **Section 1.3.7 - Stripchart Page / Tab** for more information.

The Stripchart page allows you to select and create default strip charts containing various variable associated to the operation.

7.5 Tee

The Tee operation splits one feed stream into multiple product streams with the same conditions and composition as the feed stream, and is used for simulating pipe tees and manifolds.

The dynamic Tee operation functions very similarly to the steady state Tee operation. However, the enhanced holdup model and the concept of nozzle efficiencies can be applied to the dynamic Tee. Flow reversal is also possible in the Tee depending on the pressure-flow conditions of the surrounding unit operations.

7.5.1 Tee Property View

There are two ways that you can add a Tee to your simulation:

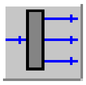
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property views property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Piping Equipment** radio button.
3. From the list of available unit operations, select **Tee**.
4. Click the **Add** button.

OR

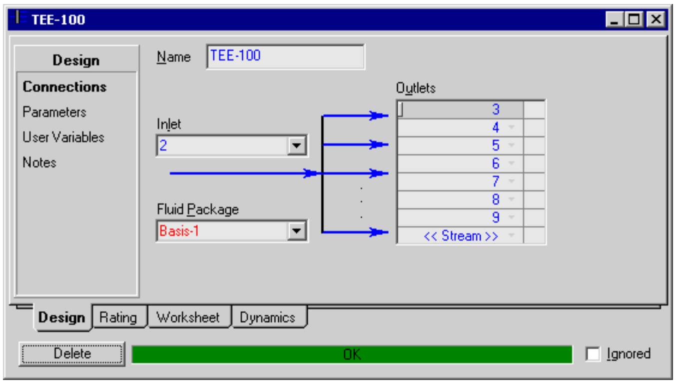
1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.
2. Double-click the **Tee** icon.


Tee icon

The Tee property view appears.

Figure 7.55



7.5.2 Design Tab

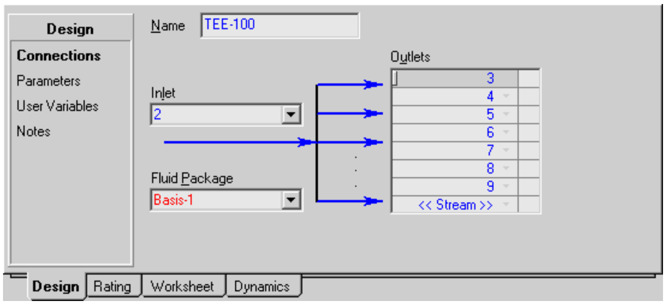
The Design tab contains the following pages:

- Connections
- Parameters
- User Variables
- Notes

Connections Page

On the Connections page, you can specify the feed stream, any number of product streams (all of which are automatically assigned the conditions and composition of the feed stream), and the fluid package associated to the Tee operation.

Figure 7.56

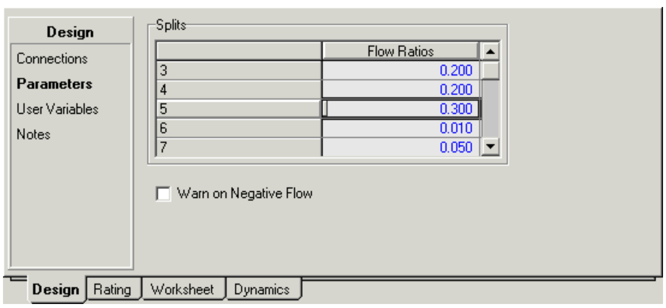


The only difference among the product streams is the flow rate, determined by the flow ratios, which you specify on the Parameters page (Steady State mode) or the outlet valve openings, which you specify on the Dynamics page (Dynamic mode).

Parameters Page

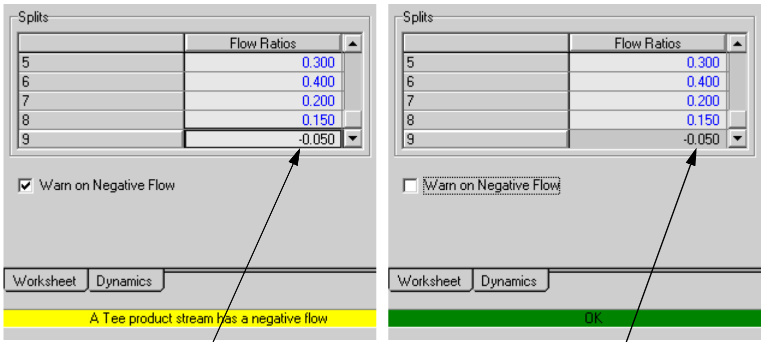
For steady state calculations, specify the desired flow ratio (the ratio of the outlet stream flow to the total inlet flow). You can toggle between ignoring or acknowledging when a negative flow occurs by selecting the **Warn on Negative Flow** checkbox.

Figure 7.57



A flow ratio is generally between 0 and 1; however, a ratio greater than one can be given. In that case, at least one of the outlet streams have a negative flow ratio and a negative flow (backflow).

Figure 7.58



With the **Warn on Negative Flow** checkbox selected, the status bar is yellow when negative flow occurs.

With the **Warn on Negative Flow** checkbox clear, the status bar is green when negative flow occurs.

For N outlet streams attached to the Tee, you must specify N-1 flow ratios. HYSYS then calculates the unknown stream flow ratio and the outlet flow rates.

$$\sum_{1=i}^{N} r_i = 1.0 \tag{7.42}$$

$$r_i = \frac{f_i}{F} \tag{7.43}$$

where:

r_i = *flow ratio of the ith stream*

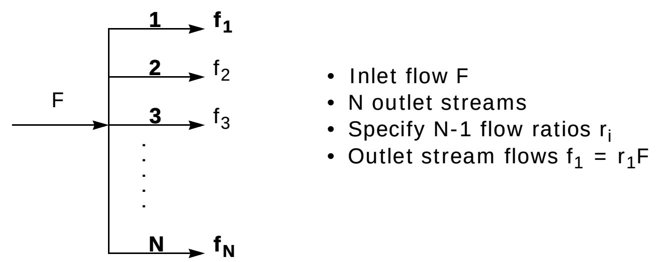
f_i = *outlet flow of the ith stream*

F = *feed flow rate*

N = *number of outlet streams*

For example, if you have four outlet streams attached to the Tee, you must give three flow ratios and HYSYS calculates the fourth.

Figure 7.59



If you switch to **Dynamic mode**, the flow ratio values do not change if the values are between 0 and 1 (they are equal to the dynamic flow fractions).

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

7.5.3 Rating tab

You need HYSYS dynamics to specify any rating information for the Tee operation.

Nozzles Page

Refer to **Section 1.3.6 - Nozzles Page** for more information.

The Nozzles page contains information regarding the elevation and diameter of the nozzles.

It is strongly recommended that the elevation of the inlet and exit nozzles are equal for this unit operation. If you want to model static head, the entire piece of equipment can be moved by modifying the Base Elevation relative to the Ground Elevation field.

7.5.4 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

The PF Specs page is relevant to dynamics cases only.

7.5.5 Dynamics Tab

The Dynamics tab contains the following pages:

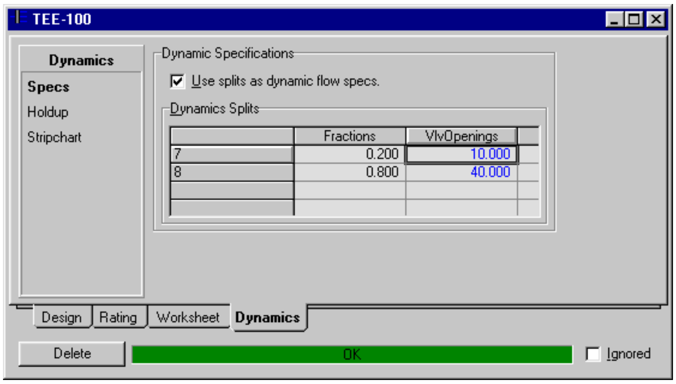
- Specs
- Holdup
- Stripchart

If you are working exclusively in Steady State mode, you are not required to change any information on the pages accessible through the Dynamic tab.

Specs Page

The dynamic specifications of the Tee can be specified on the Specs page:

Figure 7.60



In Dynamic mode, there are two specifications you can choose to characterize the Tee operation.

If the **Use Splits as Dynamic Flow Specs** checkbox is selected, the exit flows streams from the Tee are user-defined. You can define the molar flow for each exit stream by specifying the specific valve openings for each exit stream from the Tee. This situation is not recommended since the flow from the Tee is determined from split fractions and not from the surrounding pressure network of the simulation case. If this option is used, the valve opening fields should be specified all Tee exit streams. In addition a single pressure and single flow specification are required by the PF solver.

If the Use Splits as Dynamic Flow Specs checkbox is inactive, the flow rates of the exit streams are determined from the pressure network. If this option is set, the dynamic Tee acts similar to a Mixer set with the Equalize All option. The “one PF specification per flowsheet boundary stream” rule applies to the Tee operation if the Use Splits checkbox is inactive. It is strongly recommended that you clear the **Use Splits** checkbox in order to realistically model flow behaviour in your dynamic simulation case.

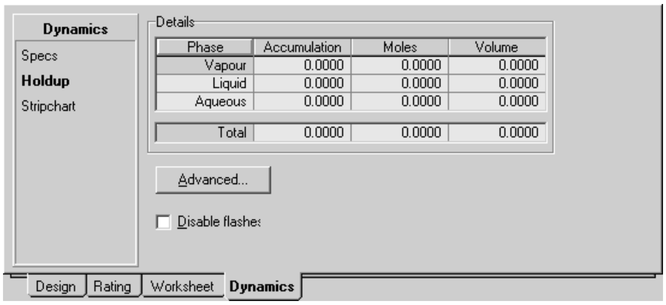
Reverse flow conditions can occur in the Tee operation if the Use Splits checkbox is inactive. If flow reverses in the Tee, it acts essentially like a dynamic Mixer with the Equalize All option. In dynamics, these two unit operations are very similar.

Holdup Page

Refer to **Section 1.3.3 - Holdup Page** for more information.

Each unit operation in HYSYS has the capacity to store material and energy. Typical Tees in actual plants usually have significantly less holdup than other unit operations in a plant. Therefore, the volume of the Tee operation in HYSYS cannot be specified and is assumed to be zero. Since there is no holdup associated with the Tee operation, the holdup's quantity and volume are shown as zero in the Holdup page.

Figure 7.61



The **Disable flashes** checkbox enables you to turn on and off the rigorous flash calculation for the tee. This feature is useful if the PFD has a very large number of tees, and you do not care whether the contents of the streams around them are fully up to date or not, or you prefer maximum speed in the simulation calculation.

- To turn off the flash calculation, select the **Disable flashes** checkbox.

  If the flash calculations are turned off, the outlet stream will still update and propagate values, but the phase fractions and temperatures may not be correct.

- To turn the flash calculation back on, clear the **Disable flashes** checkbox.

The default selection is to leave the flash calculation on.

> **HYSYS recommend that the flash calculations be left on, as in some cases disabled flash calculation can result in instabilities or unexpected outcomes, depending on what is downstream of the unit operation where the flash has been turned off. This feature should only be manipulated by advanced users.**

Stripchart Page

Refer to **Section 1.3.7 - Stripchart Page/Tab** for more information.

The Stripchart page allows you to select and create default strip charts containing various variable associated to the operation.

7.6 Valve

HYSYS performs a material and energy balance on the inlet and exit streams of the Valve operation. HYSYS performs a flash calculation based on equal material and enthalpy between the two streams. It is assumed that the Valve operation is isenthalpic.

The following is a list of variables that can be specified by the user in the Valve operation.

- Inlet temperature
- Inlet pressure
- Outlet temperature
- Outlet pressure
- Valve Pressure Drop

A total of three specifications are required before the Valve operation solves. At least one temperature specification and one pressure specification are required. HYSYS calculates the other two unknowns.

There are also a number of new features that are available with the Valve operation. The Valve is a basic building block in HYSYS dynamic cases. The new Valve operation models control valves much more realistically. The direction of flow through a Valve is dependent on the pressures of the surrounding unit operations. Like the steady state Valve, the dynamics Valve operation is isenthalpic.

Some of the new features in the Valve operation include:

- A pressure-flow specification option that realistically models flow through the valve according to the pressure network of the plant. Possible flow reversal situations can therefore be modeled.
- A pipe segment contribution that can model pressure losses caused by an attached pipe's roughness and diameter.
- A new valve equation that incorporates static head and frictional losses from the valve and/or pipe segment.
- A model incorporating Valve dynamics such as the stickiness in the valve and dynamic behaviour in the actuator.
- Different valve types such as linear, equal percentage, and quick opening valves.
- Built-in sizing features that determine valve parameters used in the valve equation.

The total valve pressure drop refers to the total pressure difference between the inlet stream pressure and the exit stream pressure. The total pressure drop across the Valve is calculated from the frictional pressure loss of the Valve, and the pressure loss from static head contributions.

7.6.1 Valve Property View

There are two ways that you can add a Valve to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.

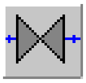
2. Click the **Piping Equipment** radio button.
3. From the list of available unit operations, select **Valve**.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.

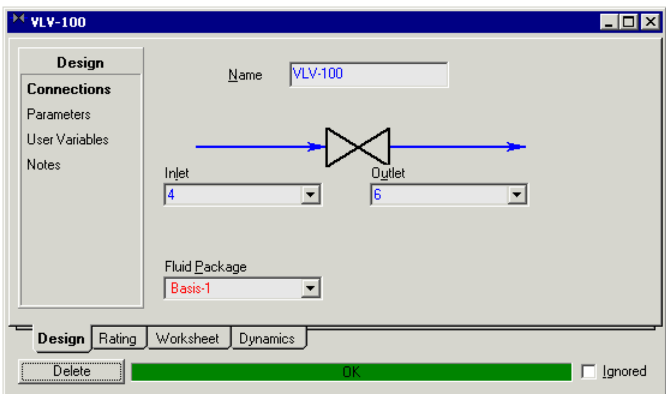
2. Double-click the **Valve** icon.



Valve icon

The Valve property view appears.

Figure 7.62



7.6.2 Design Tab

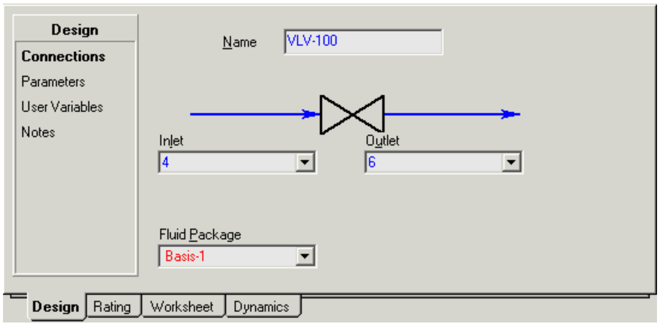
The Design tab contains the following pages:

- Connections
- Parameters
- User Variables
- Notes

Connections Page

The Connections page allows you to specify the name of the operation, as well as the inlet stream and outlet stream.

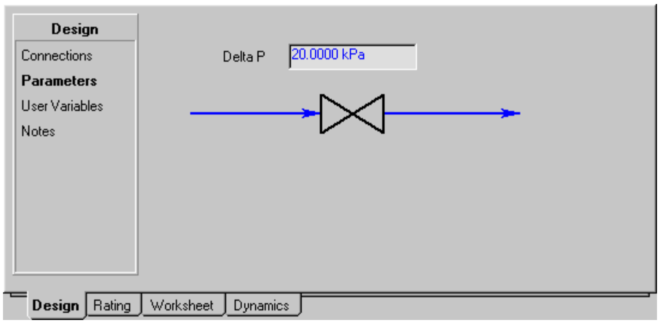
Figure 7.63



Parameters Page

The pressure drop of the Valve operation can be specified on the Parameters page.

Figure 7.64



User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

7.6.3 Rating Tab

The Rating tab contains the following pages:

- Sizing
- Nozzles
- Options

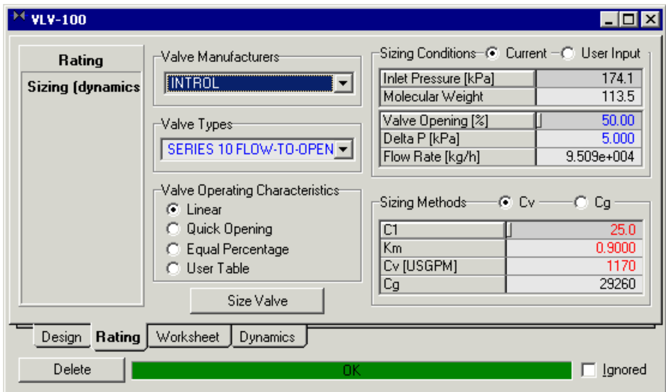
If you are working exclusively in Steady State mode, you are not required to change any information on the pages accessible through the Rating tab.

Sizing (dynamics) Page

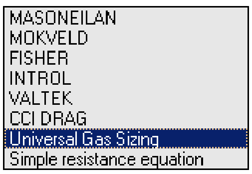
The Sizing (dynamics) page contains the following groups:

- Valve Manufacturer group
- Valve Type group
- Sizing Conditions group
- Valve Operating Characteristics group
- Sizing Methods group

Figure 7.65



Valve Manufacturer Group

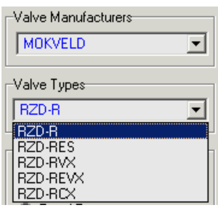


The Valve Manufacturers group contains a drop-down list that allows you to select the valve manufacturer type/equation model. The figure on the left, displays all the manufacturers available in HYSYS.

- Masoneilan
- Mokveld

- Fisher. This equation model is based on the test program results from Fisher Controls International, Inc. (developed the equation in 1963). This model accurately predicts the flow for either high or low recovery valves, for any gas and under any service conditions.
- Introl
- Valtek
- CCI Drag
- Universal Gas Sizing. This equation model is very similar to the Fisher equation. The difference between the two is the backwards compatibility when the valve has both vapour and liquid flowing through. The Universal Gas Sizing model uses the overall density for the vapour. Future versions of HYSYS will allow you to select the density model.
- Simple resistance equation. This equation model treats the flow as always being proportional to the square root of the pressure drop. No choking is modelled. This equation is often used when a simple model is desired, or if you want to calculate and update the equation constant.

Valve Type Group



Valve Type group

Some of the valve manufacturers also provide different types of valves for you to choose from. This group is only available if the manufacturer valve you selected, provide multiple valve types.

The following table lists the manufacturer valve and their associated valve type:

Valve Manufacturer	Valve Type	
Masoneilan	• DP Globe: V-Port • SP Globe: flow to open • Control Ball • Globe: contoured • Split Body: flow to open	• Split Body: flow to close • 40000,41000 Series • Camflex: flow to close • Camflex: flow to open • Butterfly (Minitork) • Gobe: flow to close
Mokveld	• RZD-R • RZD-RES • RZD-RVX	• RZD-REVX • RZD-RCX

Valve Manufacturer	Valve Type	
Introl	• Series 60A • Series 60 • Series 10 HF • Series 20 H • Series 10 HFD • Series 20 HFD	• Series 10 flow to open • Series 20 flow to open • Series 10 flow to close • Series 10 HFT • Series 20 HFT
Valtek	• Vector One 60 deg • Vector One 90 deg • Dragon Tooth	• Mark One flow to open • Mark Two flow to open • Mark One flow to close • Mark Two flow to close

Sizing Conditions Group

HYSYS uses the stream conditions provided in the Sizing Conditions group to calculate valve parameters, which are used in the valve equation.

This group contains two radio buttons and a table.

- **Current** radio button. Allows you to view and modify the current variable values for the valve sizing conditions. The current variable values are calculated based on the stream flow rate and HYSYS default values provided in the table.
- **User Input** radio button. Allows you to view and modify the variable values for the valve sizing conditions. The variable values are calculated based on the values you provide in the table.
- The table contains the following cells:

Cell	Description
Inlet Pressure	Displays the inlet pressure of the fluid flowing through the valve. This value cannot be modified.
Molecular Weight	Displays the molecular weight of the fluid flowing through the valve. This value cannot be modified.
Valve Opening	Allows you to modify the percentage opening of the valve.
Delta P	Allows you to specify the pressure difference in the valve.
Flow Rate	Allows you to modify the mass flow rate of the fluid flowing through the valve. You can only modify the mass flow rate, if you select the **User Input** radio button.

Valve Operating Characteristics Group

The Valve Operating Characteristics group contains four radio button and a button.

Refer to **Theory** section for more information about each calculation method.

Object	Description
Linear radio button	Allows you to select Linear method to calculate the valve size.
Quick Opening radio button	Allow you to select Quick Opening method to calculate the valve size.
Equal Percentage radio button	Allows you to select Equal Percentage method to calculate the valve size.
User Table radio button	Allows you to specify the valve characteristics curve values used to calculate the valve size.
View button	Allows you access to the Characteristics Curve property view. This button is only available if you select **User Table** button.

To select the method to characterize the valve:

1. In the Valve Operating Characteristics group, select the method you want to use by clicking the appropriate radio button: **Linear**, **Quick Opening**, **Equal Percentage**, and **User Table**.

Refer to **Sizing Methods Group** section for more information.

If you select **Linear**, **Quick Opening**, or **Equal Percentage**:

2. In the Sizing Methods group, select C_v or C_g radio button and specify the parameter values in the appropriate cells.

Refer to **Characteristics Curve Property View** section for more information.

If you select **User Table**:

3. In the Valve Operating Characteristics group, click the **View** button. The valve Characteristics Curve property view appears.
4. In the **Lift (% of max)** column, specify the percentage of the valve opening. This percentage value is based on the maximum valve opening.
5. In the **Flow (% of max)** column, specify the flow rate percentage. This percentage value is based on the maximum fluid flow rate through the valve.

Characteristics Curve Property View

The Characteristics Curve property view contains a table, two buttons, and a plot:

Object	Description
Lift (% of max) column	Allows you to specify the valve stem position percentage value. The values are limited to a range of 0 to 100.
Flow (% of max) column	Allows you to specify the percentage flow rate associated to the valve stem position. The values are limited to a range of 0 to 100.
Erase Selected button	Allows you to erase the selected percentage stem position and percentage flow rate values.
Erase All button	Allows you to erase all the values in the Curve Information table.
Plot	Displays the characteristics curve based on the values specified in the Curve Information table.

The table must contain at least three points to calculate the characteristics curve of the valve operation.

If the values in the Curve Information table is not valid and the User Table method has been selected, then you will not be able to run the valve in dynamic mode.

When you first open the Characteristics Curve property view, the **Lift (% of max)** and **Flow (% of max)** columns contain two mandatory values (0 and 1.0) and three default values (0.25, 0.5, and 0.75).

Theory

In HYSYS there are four different methods to characterize the control valve. All four methods use the following parameters to calculate the control valve:

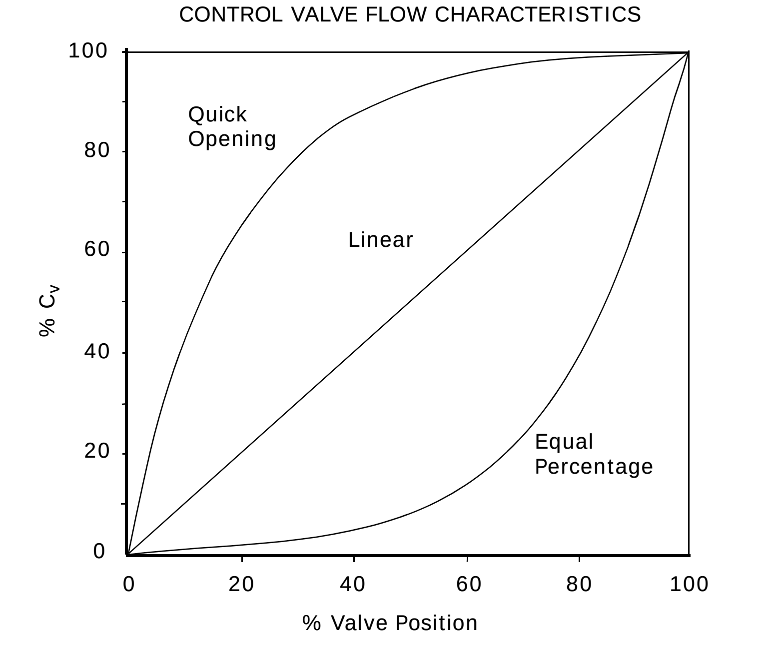
The flow rate through a control valve depends on the actual valve position. If the flow can be expressed in terms of $\%C_v$,(0% representing no flow conditions and 100% representing the maximum flow conditions) then the valve characteristics of a control valve is defined as the dependence on the quantity of $\%C_V$ as a function of the actual valve percent opening.

Three methods use equation models based on three different types of valves:

Valve Type	Description
Linear	A control valve with linear valve characteristics has a flow which is directly proportional to the valve % opening. The mathematical relationship of Cv (%) and Valve Position (%) for Linear valve is as follows: $$\%C_v = (\%\text{Valve Opening}) \quad (7.44)$$
Quick Opening	A control valve with quick opening valve characteristics obtains larger flows initially at lower valve openings. As the valve opens further, the flow increases at a smaller rate. The mathematical relationship of Cv (%) and Valve Position (%) for Quick Opening valve is as follows: $$\%C_v = (\%\text{Valve Opening})^{0.5} \quad (7.45)$$
Equal Percentage	A control valve with equal percentage valve characteristics initially obtains very small flows at lower valve openings. However, the flow increases rapidly as the valve opens to its full position. The mathematical relationship of Cv (%) and Valve Position (%) for Equal Percentage valve is as follows: $$\%C_v = (\%\text{Valve Opening})^3 \quad (7.46)$$

The valve characteristics are shown graphically in the figure below.

Figure 7.66



The forth method used a table filled with values of the valve stem positions and the associate percentage flow capacity characteristic curves to calculate the operating characteristics. In other words, the forth method does not rely on an equation to predict the control valve flow characteristics, instead it relies on values supplied by the user.

The calculation is done by using simple quadratic interpolation from the three points in the table closest to the current stem position.

Sizing Methods Group

The Sizing Methods group contains two radio button, a button, and a table:

Object	Description
Cv radio button	Allows you to select C_v as the parameter to manipulate the resistance equation in the flow calculation.
Cg radio button	Allows you to select C_g as the parameter to manipulate the resistance equation in the flow calculation.
C1 cell	Allows you to specify the ratio value of C_g/C_v.
Km cell	Allows you to specify the pressure recovery coefficient. This coefficient is used in choked liquid flow calculations
Cv cell	Allows you to specify the fluid flow rate (C_v) value. This cell is only active if you select **Cv** radio button.
Cg cell	Allows you to specify the gas sizing coefficient. This cell is only active if you select **Cg** radio button.
k cell	Allows you to specify the *k* value for the Simple Resistance equation method. This cell is only available if you select **Simple resistance equation** in the Valve Manufacturer group
Size Valve button	Allows you to specify a single valve parameter, while HYSYS calculates the remaining parameter values based on the stream and valve conditions (the conditions are taken from the Sizing Conditions group). HYSYS provides a C_1 default value of 25.

Theory

The sizing calculation method is the same for all valve manufacturers and types, with the exception of the **Simple Resistance Equation**.

The difference between the manufacturers and types is the equations and constants used to calculate the flow rate within the valve. All valve manufacturers and types have C_v and C_g methods to calculate flow rate.

The following equations provide an example of the sizing method calculation for Universal Gas Sizing valve:

- The C_v *and* C_g methods calculate the vapour flow through the valve using the following equation:

$$(lb/\ hr) = \upsilon_{fracfac} 1.06 C_g \sqrt{\rho(lb/\ ft^3) \times P_1} \times \sin\left(\frac{59.64}{C_1}\sqrt{1 - \frac{P_2}{P_1}} \times cp_{fa}\right. \tag{7.47}$$

where:

$$C_1 = \frac{C_g}{C_v} \tag{7.48}$$

$$Km = 0.001434 C_1 \tag{7.49}$$

$$cp_{fac} = \sqrt{\frac{0.4839}{1 - \left(\frac{2}{1+\gamma}\right)^{\left(\frac{\gamma}{\gamma - 1}\right)}}} \tag{7.50}$$

$$\gamma = C_p /\ C_v \tag{7.51}$$

P_1 = *pressure of the inlet stream*

P_2 = *pressure of the exit stream without static head contributions*

$\upsilon_{fracfac}$ = *1, outlet molar vapour fraction* $v_{\mathrm{frac}} > 0.1$
= *0, outlet molar vapour fraction* $v_{\mathrm{frac}} = 0$
= $v_{\mathrm{frac}}/0.1$, *otherwise*

- For the liquid flow through the valve, the equation is as follows:

$$f(lb/\ hr) = \upsilon_{fracfac} \times 63.338 \times C_v \times \sqrt{\rho(lb/\ ft^3)} \times \sqrt{P_1 - P_2} \tag{7.52}$$

HYSYS reports the full C_v (at 100% open, which remains fixed) plus the valve opening. If the Valve is 100% open then you get a smaller Valve than if the Valve was only 50% open for the same conditions. This is just one way of sizing a Valve as some sources report an effective C_v (varies with the valve opening) versus the value opening.

The above equations are not rigorous for two-phase flow.

Simple Resistance Equation

If the *Simple Resistance Equation* is chosen, you can either:

- Specify *k* value.
- Have *k* calculated from the stream and valve conditions displayed in the Sizing Conditions group. To calculate *k* value, specify the required variable values and click the **Size Valve** button.

The *Simple Resistance Equation* method calculates the flow through the Valve using the following equation:

$$f = k\sqrt{density \times valveopening \times (P_1 - P_2)} \tag{7.53}$$

The general valve flow equation uses the pressure drop across the Valve without any static head contributions. The quantity, P_1 - P_2, is defined as the frictional pressure loss, which is used in the valve sizing calculation. The valve opening term is dependant on the type of Valve and the percentage that it is open. For a linear valve:

$$valveopening = \left(\frac{\%\text{ valve open}}{100}\right)^2 \tag{7.54}$$

The inverse relationship between the percentage of valve opening, and C_g can be shown as follows:

$$(\%\text{ valve open}) \times C_g = Flow \tag{7.55}$$

When the valve size is fixed, the percentage of valve opening increases with the flow through the valve. However, when sizing a valve, the C_g is not fixed. The C_g is inversely dependent on the flow, and the percentage of valve opening.

Nozzles Page

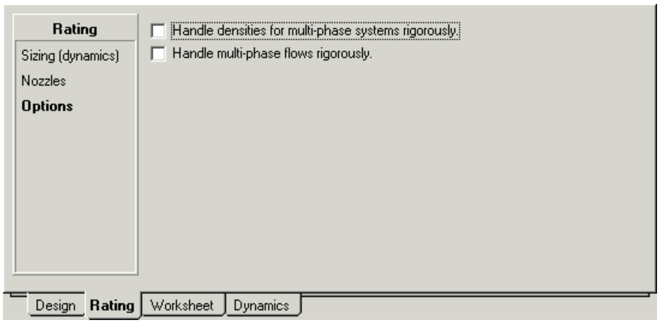
Refer to **Section 1.3.6 - Nozzles Page** for more information.

The Nozzles page contains information regarding the elevation and diameter of the nozzles.

Options Page

The Options page enables you to select the method to handle multiphase streams.

Figure 7.67



- The **Handle densities for multi-phase systems rigorously** checkbox enables the valve to use the phase densities to calculate flow rate for the vapour and liquid. If the option was not selected the overall density is used.
- The **Handle multi-phase flows rigorously** checkbox enables the valve to use rigorous calculation method and obtain better accuracy of the flow rates and pressure drop for both vapour and liquid flow. The calculations have been improved to be consistent with the Fisher calculations for multiphase systems.

7.6.4 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

The PF Specs page is relevant to dynamics cases only.

7.6.5 Dynamics Tab

The Dynamics tab contains the following pages:

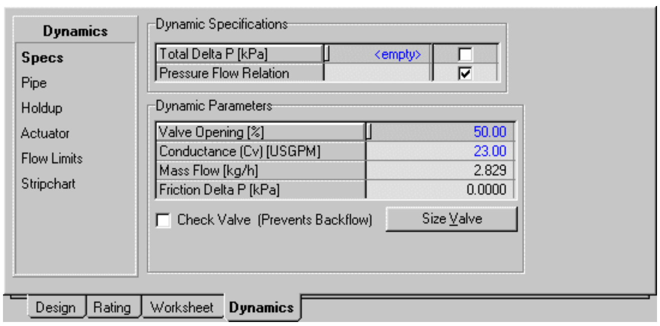
- Specs
- Pipe
- Holdup
- Actuator
- Flow Limits
- Stripchart

If you are working exclusively in Steady State mode, you are not required to change any information on the pages accessible through the Dynamics tab.

Specs Page

The dynamic specifications and parameters of the Valve can be specified on the Specs page.

Figure 7.68



Dynamic Specifications

If the **Total Delta P** checkbox is selected, a set pressure drop is assumed across the Valve operation. With this specification, the flow and the pressure of either the inlet or exit stream must be specified or calculated from other operations in the flowsheet. The flow through the Valve is not dependent on the pressure drop across the Valve.

In Dynamic mode, there are two possible dynamic specifications you can choose to characterize the Valve operation.

If the **Pressure Flow Relation** checkbox is selected, two of the following pressure-flow specifications must either be specified or calculated by the other unit operations in the flowsheet:

- Inlet Stream Pressure
- Exit Stream Pressure
- Flow through the Valve

The flow rate through the Valve is calculated from the valve equation, and the pressure of the streams entering and exiting the Valve.

In dynamics, the suggested mode of operation for the Valve is the Pressure Flow specification. The pressure drop option is provided for steady state compatibility mostly, and to allow difficult simulations to converge more easily. However, it usually is not a sensible specification since it allows a pressure drop to exist with zero flow.

Dynamic Parameters

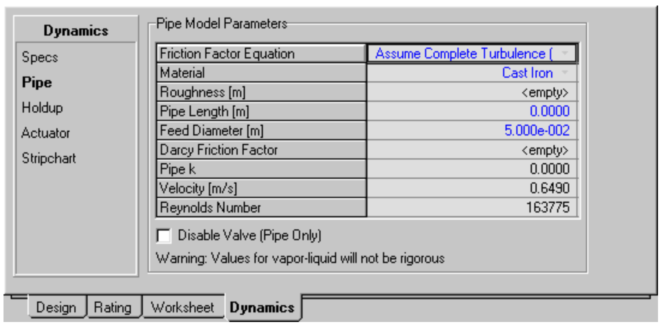
The Dynamic Parameters group lists the same stream and valve conditions required to size the Valve as in the Sizing Conditions group. The Valve Opening % and the Conductance (Cv or k) appear and can be modified in the section. The conductance of the Valve can be calculated by clicking the Size Valve button.

The **Check Valve** checkbox can be selected if you do not want flow reversal to occur in the Valve.

Pipe Page

The Valve module supports a pipe contribution in the pressure flow equation.

Figure 7.69



A pipe contribution DOES NOT contribute to any holdup volume. You have to enter the holdup separately in the Holdup Page.

The pipe calculations for a valve are not rigorous for multiphase flow and are only approximations.

This can be used to model a pipe segment in the feed to the Valve, but it is also possible to disable the valve contribution and have the Valve unit operation act as a simple pipe segment only. The pressure flow specification has to be enabled in order for the pipe segment to be modeled.

The following pipe modeling parameters appear in this section:

- Friction Factor Equation
- Material
- Roughness
- Pipe length
- Feed diameter
- Darcy friction factor
- Pipe k
- Velocity
- Reynolds number

Friction Factor

The Friction Factor Equation option allows you to choose between two different equations:

- Assume Complete Turbulence (f is fixed)

  Assume Complete Turbulence is the default equation, and the calculation is fast and simple. This method calculates the friction factor once and uses that value irrespective of the Reynolds number (the calculated friction factor value is not correct if the flow is laminar).
- Full-Range Churchill (covers all flow regimes)[1,7]

  The Full-Range Churchill method calculates the friction factor as a function of the Reynolds number. This method is slower but calculates a unique friction factor for the turbulent, lamiar, and transtitional regions. If the flow through the Valve is too low HYSYS uses a low limit of 10 for the Reynolds number.

HYSYS suggests a typical pipe roughness if the pipe material is specified. The pipe roughness may also be directly specified. The feed diameter and pipe length must be specified as well. These specifications are used to determine the Darcy friction factor.

The friction factor is calculated as follows:

Assume Complete Turbulence equation:

$$\frac{1}{\sqrt{f_{friction}}} = 2.457\ln\left(\frac{3.707D}{\varepsilon}\right) \tag{7.56}$$

Full-Range Churchill equation:

$$f_{friction} = \left[\left(\frac{8}{Re}\right)^{12} + \frac{1}{(A+B)^{1.5}}\right]^{1/12} \tag{7.57}$$

$$f_{Darcy} = 8 \times f_{friction} \tag{7.58}$$

where:

f_{Darcy} = *Darcy friction factor*

D = *pipe diameter*

ε = *pipe roughness*

$$A = \left[2.457\ln\frac{1}{\left(\frac{7}{Re}\right)^{0.9} + 0.27\left(\frac{\varepsilon}{D}\right)}\right]^{16}$$

$$B = \left(\frac{37530}{Re}\right)^{16}$$

A pipe k-value is calculated from the Darcy friction factor and the pipe diameter. The pipe k value is incorporated into the general valve equation.

Notice that this pipe k is independent of the flow rate or pressure of the fluid in the Valve.

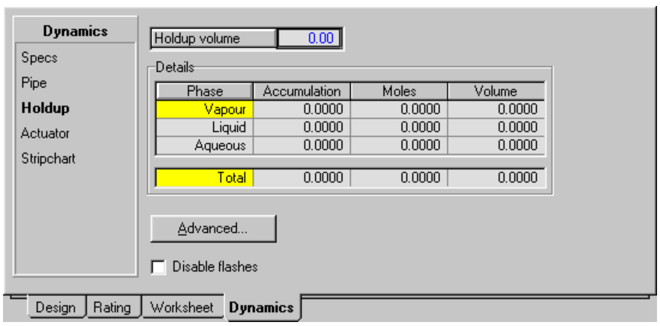
The pipe segments only calculate frictional losses. They do not automatically calculate holdup volume. You must enter this on the Holdup page of the Dynamics tab.

Holdup Page

Refer to **Section 1.3.3 - Holdup Page** for more information.

The Holdup page contains information regarding the holdup properties, composition, and amount.

Figure 7.70



Each unit operation in HYSYS has the capacity to store material and energy. Typical Valves usually have significantly less holdup than other unit operations in a plant. Therefore, the volume of the Valve operation in HYSYS is defaulted to be zero.

The Holdup page allows you to:

- Specify a non-zero value (in the **Holdup Volume** field) for the holdup volume in the Valve.

Refer to **Section 2.4 - Integrator** in the **HYSYS Dynamic Modeling** guide for more information.

The HYSYS Dynamics license is required and the Access Fidelity license options checkbox selected in the Options tab of the Integrator property view, to set a non-zero holdup.

- Disable any flashes that may occur in the Valve.

Holdup occurs after the valve, whereas the pipe contribution occurs before the valve.

The **Disable flashes** checkbox enables you to turn on and off the rigorous flash calculation for the valve. This feature is useful if the PFD has a very large number of valves, and you do not care whether the contents of the streams around them are fully up to date or not, or you prefer maximum speed in the simulation calculation.

- To turn off the flash calculation, select the **Disable flashes** checkbox.

  If the flash calculations are turned off, the outlet stream will still update and propagate values, but the phase fractions and temperatures may not be correct.
- To turn the flash calculation back on, clear the **Disable flashes** checkbox.

The default selection is to leave the flash calculation on.

HYSYS recommend that the flash calculations be left on, as in some cases disabled flash calculation can result in instabilities or unexpected outcomes, depending on what is downstream of the unit operation where the flash has been turned off. This feature should only be manipulated by advanced users.

Actuator Page

Refer to **Section 1.6.3 - Control Valve Actuator** in the **HYSYS Dynamic Modeling** guide for more information.

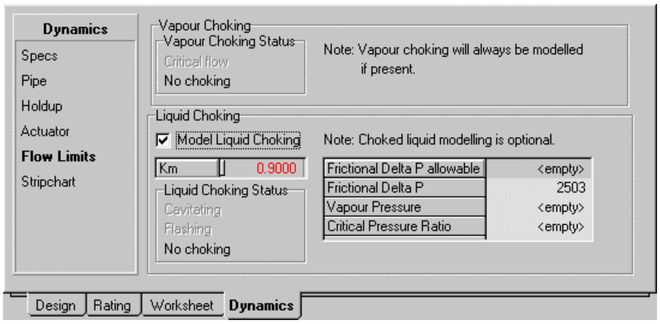
The Actuator page allows you to model valve dynamics in the Valve operation. The HYSYS Dynamics license is required to use the Actuator features found on this page.

Flow Limits Page

The Flow Limits page allows you to monitor the status of the vapour and liquid flow passing through the valve. The page consists of two groups:

- Vapour Choking
- Liquid Choking

Figure 7.71



Vapour Choking Group

By default, the Vapour Choking status is always monitored whenever it is applicable. You can view the current condition of the vapour flow in the Vapour Choking Status group. The active status is shown in black whereas the inactive status is greyed out. The two statuses available are Critical Flow and No Choking.

Critical flow or vapour choking refers to the crowding condition when the gas flowing through the valve has exceeded the designed limit and reaches the sonic velocity.

Critical flow is calculated by the Fisher equation for gases.

$$f = V_{frac} \times 1.06 \times C_g \times \sqrt{\rho} \times \sqrt{P_1} \times \sin\left(\frac{59.64}{C_1} \times \sqrt{1 - \frac{P_2}{P_1}} \times C_{p_{fac}}\right) \qquad (7.59)$$

where:

f = *flow (lb/hr)*

V_{frac} = *vapour fraction*

C_g = *Fisher's valve vapour coefficient*

C_1 = *critical flow factor,* C_g/C_v *(between 33 - 38)*

C_{pfac} = *theoretical correction factor for the ratio of specific heats*

ρ = *density* (lb/ft^3)

P_1 = *pressure at valve inlet (psia)*

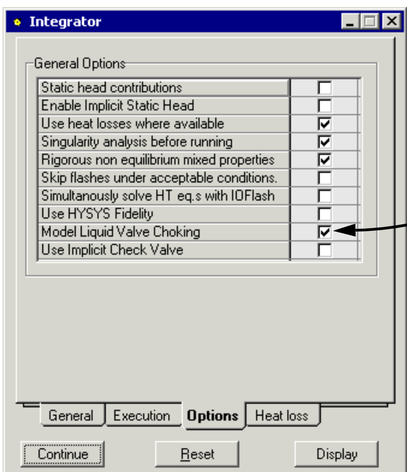
P_2 = *pressure at valve outlet (psia)*

As the Sine function approaches to 1, the flow through the valve becomes choked and under this condition, the vapour through the valve undergoes critical flow.

Liquid Choking Group

Liquid choked modeling is optional in HYSYS. You can turn it on or off for the associated valve by selecting the **Model Liquid Choking** checkbox.

Figure 7.72



You can also select the **Model Liquid Choking** checkbox on the **Options** tab of the Integrator property view.

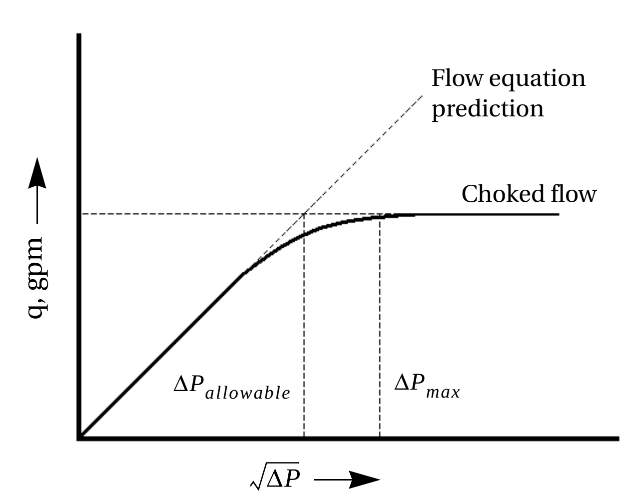
There are three flow conditions in the Liquid Choking Status group:

- **No choking**. Normal flow condition. An increase in pressure drop across the valve results in an increased flow. The No choking condition holds for a limited range.
- **Flashing**. When the pressure of the valve outlet falls below the vapour pressure of the liquid.
- **Cavitating**. When the pressure of valve outlet raises above the vapour pressure of the liquid.

The presence of these flow conditions can significantly affect the valve performance and the overall process. During flashing, liquid starts to vapourize, and the change in phase from liquid to vapour causes bubbles to form. This creates congestion across the valve (liquid choking), and the flow is severely limited. At this point, increase in pressure drop will not result in increased flow. When cavitation occurs, the pressure of the liquid recovers and raises above its vapour pressure. This causes vapour bubbles to collapse and burst, producing a great amount of noise and vibrations that can damage the valve.

Since the regular Fisher liquid flow equation does not predict liquid choked flow, a different equation is used to take into account the effects of flashing and cavitation.

Figure 7.73



K_m (the pressure recovery coefficient) predicts flashing and cavitation for a valve. By default, the pressure recovery coefficient is set at a conservative value of 0.9. You can specify the K_m value to adjust the flow condition.

The liquid choked-flow condition is shown in **Figure 7.73**. As pressure drop increases, the liquid flow becomes choked. The allowable pressure drop ($\Delta P_{allowable}$) indicates when the liquid choked-flow occurs and it is defined as:

$$\Delta P_{allowable} = K_m(P_1 - r_c P_v) \tag{7.60}$$

where:

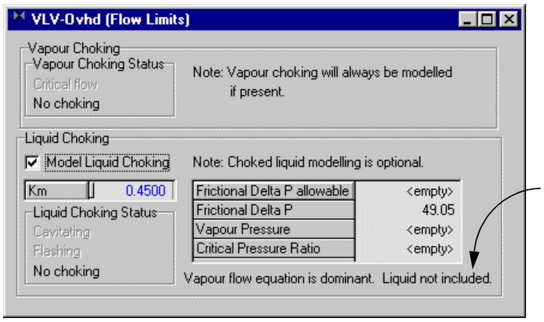
K_m = *pressure recovery value*

r_c = *critical pressure ratio*

P_v = *vapour pressure of liquid*

The values of these parameters are displayed in the table. The Frictional Delta P shows the current pressure drop across the valve. As you adjust the pressure recovery coefficient, the Frictional Delta P allowable changes according to **Equation (7.60)**. The Liquid Vapour pressure and the Critical Pressure Ratio are also displayed for reference.

Figure 7.74

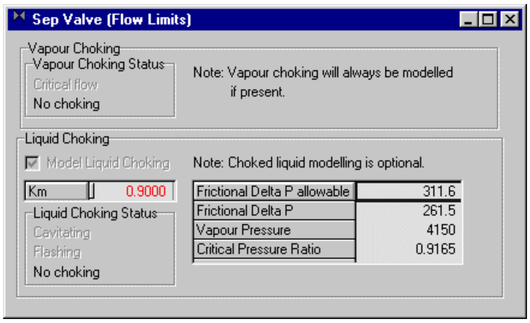


If the flow is vapour dominant, HYSYS displays this message even if the **Model Liquid Choking** checkbox is selected.

If the Frictional Delta P allowable is below the Frictional Delta P, then cavitation occurs.

No message is displayed when the flow consists of vapour and liquid.

Figure 7.75



Stripchart Page

Refer to **Section 1.3.7 - Stripchart Page / Tab** for more information.

The Stripchart page allows you to select and create default strip charts containing various variable associated to the operation.

7.7 References

[1] American Gas Association on the "Engineering Data Book SI Volume II", 11th edition, GPSA, 1998, page 17.

[2] Aziz, K., Govier, G.A., and Fogarasi, M., *Pressure Drop in Wells Producing Oil and Gas*, Journal of Canadian Petroleum Technology, July-September 1972, pp 38-48.

[3] Baxendell, P.B., and Thomas, R., *The Calculation of Pressure Gradients in High-Rate Flowing Wells*, J. Pet. Tech., October 1961, pp 1023-1028.

[4] Beggs, H.D., and Brill, J.P., *A Study of Two-Phase Flow in Inclined Pipes*, J. Petrol. Technol., p. 607, May (1973).

[5] Brill, J.P., and Beggs, H.D., Two Phase Flow in Pipes, Sixth Ed, July 1989.

[6] Brill, J.P., and Mukherjee, H., *Multiphase Flow in Wells*, SPE Monograph, Volume 17.

[7] Churchill, S. W., Chem. Eng., 84(24), (1977), 91.

[8] Duns, H.Jr., and Ros, N.C.J., *Vertical Flow of Gas and Liquid Mixtures in Wells*, 6th World Petroleum Congress, Frankfurt, June 1963, pp 451-465.

[9] Eckert, E.R.G. & Drake, R.M., "Analysis of heat and mass transfer", HTFS Reference Number 60167, 1972.

[10] Gregory, G.A., Mandhane, J. and Aziz, K., *Some Design Considerations for Two-Phase Flow in Pipes*, J. Can. Petrol. Technol., Jan. - Mar. (1975).

[11] Hagedorn, A.R., and Brown, K.E., *Experimental Study of Pressure Gradients Occurring During Continuous Two-Phase Flow in Small-Diameter Vertical Conduits*, Journal of Petroleum Technology, April 1965, pp 475-484.

[12] HTFS Handbook, Volume 2, Methods TP3, TM4, TM5

[13] HTFS Handbook, Volume 2, Two Phase Flow.

[14] Multiphase Flow and Subsea Separation. Special report by Smith Rea Energy Associates and UKAEA, April 1989.

[15] Orkisewski, J., *Predicting Two-Phase Pressure Drops in Vertical Pipe*, Journal of Petroleum Technology, June 1967, pp 829-839.

[16] Poettmann, F.H., and Carpenter, P.G., *The Multiphase Flow of Gas, Oil and Water Through Vertical Flow Strings with Application to the Design of Gas-Lift Installations*, Drill and Prod. Practice, API, pp. 257-317, March 1952.

[17] Smith, R.A. et al, *Two Phase Pressure Drop*, HTFS Design Report 28 (Revised), 1981 (8 parts, 2 Appendices).

[18] Tengesdal, J.Ø, Sarica, C., Schmidt, Z., and Doty, D., *A Mechanistic Model for Predicting Pressure Drop in Vertical Upward Two-Phase Flow*, Journal of Energy Resources Technology, March 1999, Vol 121.

[19] Watson, M., *The modelling of slug flow properties*, 10th International Conference Multiphase '01, Cannes, France, 13-15 June 2001.

8 Reactor Operations

8.1 CSTR/General Reactors

With the exception of the Plug Flow Reactor (PFR), all of the reactor operations share the same basic property view. The primary differences are the functions of the reaction type (conversion, kinetic, equilibrium, heterogeneous catalytic or simple rate) associated with each reactor. As opposed to a separator or general reactor with an attached reaction set, specific reactor operations can only support one particular reaction type. For instance, a conversion reactor only functions properly with conversion reactions attached. If you try to attach an equilibrium or a kinetic reaction to a conversion reactor, an error message appears. The GIBBS reactor is unique in that it can function with or without a reaction set.

You have a great deal of flexibility in defining and grouping reactions. You can:

- Define the reactions inside the Basis Manager, group them into a set and then attach the set to your reactor.
- Create reactions in the Reaction Package in the main flowsheet, group them into a set, and attach the set to the reactor.
- Create reactions and reaction sets in the Basis Environment and make changes in the Main Environment's Reaction Package.

Refer to **Chapter 5 - Reactions** of the **HYSYS Simulation Basis** guide or **Section 5.3 - Reaction Package** of the **HYSYS User Guide** for details on installing reactions and Reaction Sets.

Regardless of the approach, the reactions you define are visible to the entire flowsheet. In other words, a reaction set can be attached to more than one reactor.

However, there are some subtleties of which you must be aware. When you make a modification to a reaction via a reactor, the change is only seen locally, in that particular reactor. Modifications made to a reaction in the Basis Environment or in the Reaction Package are automatically reflected in every reactor using the reaction set, provided you have not made changes locally. Local changes are always retained.

To override local changes and return the global parameters to a reaction, you must press the **DELETE** key when the cursor is in the cell which contains the local change.

To remove local changes, select the appropriate cell and press the **DELETE** key.

The four reactors which share common property views include:

- CSTR (Continuous-Stirred Tank Reactor)
- GIBBS Reactor
- Equilibrium Reactor
- Conversion Reactor
- Yield Shift Reactor

The last four reactors are referred to as General Reactors. In order to avoid redundancy, CSTR, Gibbs, Equilibrium, and Conversion reactor operations are discussed co-currently. In areas of the property view where there are differences, such as the Reactions tab, the differences are clearly noted.

The Yield Shift and PFR have a different property view from the other reactors. As a result it is discussed in **Section 8.3 - Yield Shift Reactor** and **Section 8.4 - Plug Flow Reactor (PFR)**.

8.1.1 Adding a CSTR/General Reactors

There are two ways that you can add a reactor operation to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

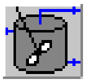
   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Reactors** radio button.
3. From the list of available unit operations, select the reactor type you want to add: Cont. Stirred Tank Reactor, Conversion Reactor, Equilibrium Reactor, Gibbs Reactor, or Yield Shift Reactor.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.

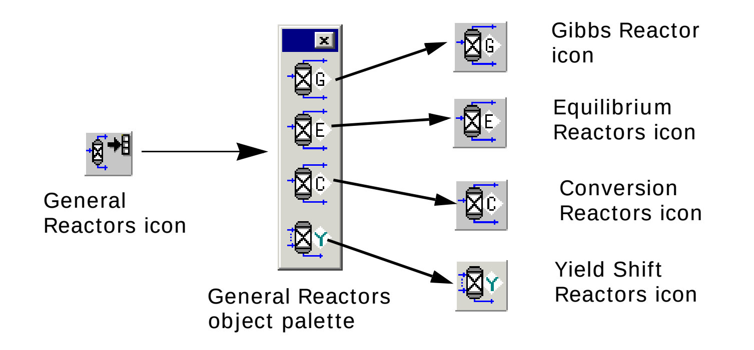
2. Do one of the following:



CSTR icon

   - For continuous-stirred tank reactor, double-click the **CSTR** icon. The CSTR property view appears.
   - For Conversion, Equilibrium, Gibbs, and Yield Shift reactors, click the **General Reactors** icon to open the General Reactors object palette.

Figure 8.1



   In the General Reactors object palette, double-click on the appropriate reactor operation icon.

The property view for the selected Reactor operation appears.

8.2 CSTR/General Reactors Property View

The CSTR and General Reactors property view contains the following tabs:

- Design
- Reactions
- Rating
- Worksheet
- Dynamics

8.2.1 Design Tab

The Design tab contains several pages, which are briefly described in the table below.

Page	Description
Connections	Connects the feed, product, and energy streams to the reactor. For more information, refer to the section below.
Parameters	Sets heat transfer and pressure drop parameters for the reactor.
User Variables	Enables you to create and implement your own user variables for the current operation.
Notes	Allows you to add relevant comments which are exclusively associated with the unit operation.

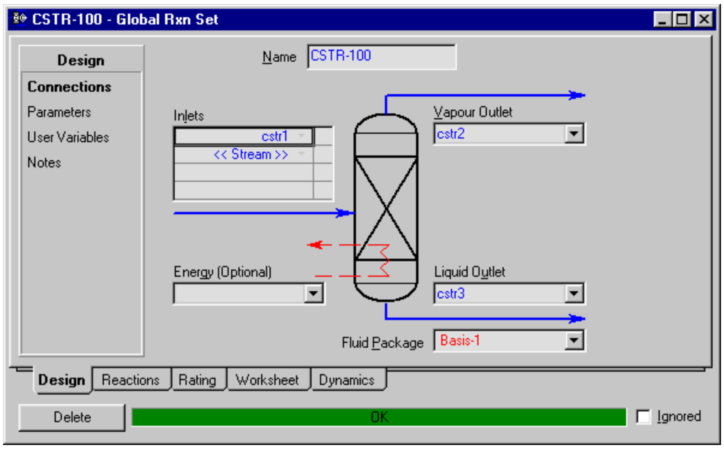
For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

For more information on the Notes page, refer to **Section 1.3.5 - Notes Page / Tab**.

Connections Page

The Connections page, is the same for both the CSTR and the General Reactors.

Figure 8.2



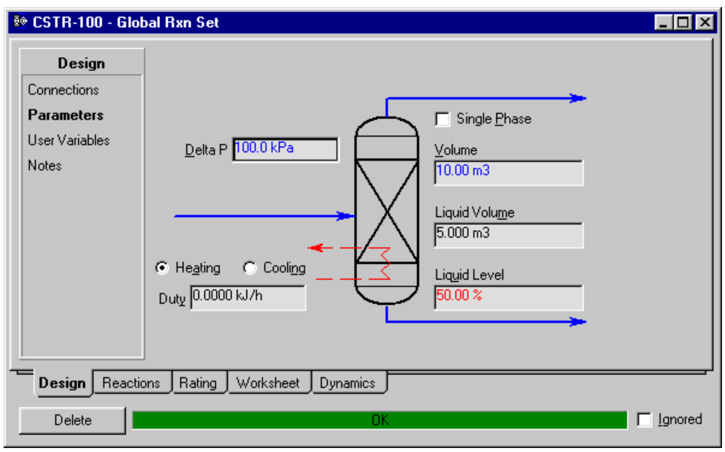
The Connections page consists of the following objects described in the table below.

Object	Input Required
Name	Contains the name of the reactor. You can edit the name of the reactor at any time by typing in a new name in the Name field.
Inlets / Feed Streams	Connects a single feed or multiple feed streams to the reactor. You can either type in the name of the stream or if you have pre-defined your stream select it from the drop-down list.
Vapour Outlet	Connects the vapour product stream to the reactor. You can either type in the name of the stream or if you have pre-defined your stream select it from the drop-down list. At least one product stream is required.
Liquid Outlet / Product Stream	Connects the liquid product stream to the reactor. You can either type in the name of the stream or if you have pre-defined your stream select it from the drop-down list.
Energy (Optional)	Connects or creates an energy stream if one is required for the operation.
Fluid Package	Enables you to select a fluid package to be associated to the reactor.

Parameters Page

The Parameters page allows you to specify the pressure drop, vessel volume, duty, and solving behaviour.

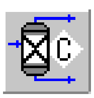
Figure 8.3



Object	Description
Delta P / Pressure Drop	Contains the pressure drop across the vessel. The pressure drop is defined as: $$\Delta P = P_{feed} - P_v = P_{feed} - P_l \qquad P = P_v = P_l \tag{8.1}$$ *where:* P = *vessel pressure* P_v = *pressure of vapour product stream* P_l = *pressure of liquid product stream* P_{feed} = *pressure of feed stream (assumed to be the lowest pressure of all the feed streams)* ΔP = *pressure drop in vessel (Delta P)* The default pressure drop across the vessel is zero. The vessel pressure is used in the reaction calculations.
Duty	If you have attached an energy stream, you can specify whether it is to be used for heating or for cooling by selecting the appropriate radio button. You also have a choice of specifying the applied duty, or having HYSYS calculate the duty. For the latter case, you must specify an outlet temperature for a reactor product stream. The steady state Reactor energy balance is defined below: $$Duty = H_{vapour} + H_{liquid} - H_{feed} \tag{8.2}$$ *where:* *Duty = heating (+ve) or cooling (-ve) by the optional energy stream* H_{vapour} = *heat flow of the vapour product stream* H_{liquid} = *heat flow of the liquid product stream* H_{feed} = *heat flow of the feed stream(s)* The enthalpy basis used by HYSYS is equal to the ideal gas enthalpy of formation at 25°C and 1 atm. As a result, the heat of reaction calculation is amalgamated into any product/reactant enthalpy difference.
Heating / Cooling	If you change from Heating to Cooling (or vice versa), the magnitude of the energy stream does not change. However, the sign changes in the energy balance. For Heating, the duty is added. For Cooling, the duty is subtracted.

Object	Description
Volume	The total volume of the vessel and is user specified. While not necessarily required for solving Conversion, GIBBS or Equilibrium reactors in Steady State mode, this value must be entered for CSTR. The vessel volume, together with the liquid level set point, define the amount of holdup in the vessel. The amount of liquid volume or holdup in the vessel at any time is given by the following expression: $Holdup = Vessel\ Volume \times \frac{PV(\%Full)}{100}$ (8.3) *where:* *PV(%Full) = liquid level in the vessel* The vessel volume is necessary when modeling reactors in steady state, as it determines the residence time.
Liquid Level	Displays the liquid level of the reactor expressed as a percentage of the Full Vessel Volume.
Liquid Volume	Not set by the user, this value is calculated from the product of the volume (vessel volume) and liquid level fraction. It is only active when the Volume field contains a valid entry.
Act as a Separator When Cannot Solve	Only available for Conversion and Equilibrium reactors, this option allows you to operate the reactor as a simple 2 phase separator whenever the reactor does not solve.
Single Phase	Allows you to specify a single phase reaction. Otherwise HYSYS considers it a vapour-liquid reaction.
Type	Only available for the Gibbs reactor, you have two options for the type of reactor you want: • **Separator**. A two phase Gibbs Reactor. • **Three Phase**. A three phase Gibbs Reactor.

8.2.2 Conversion Reactor Reactions Tab



Conversion Reactor icon

The Conversion Reactor is a vessel in which conversion reactions are performed. You can only attach reaction sets that contain conversion reactions. Each reaction in the set proceeds until the specified conversion is attained or until a limiting reactant is depleted.

Refer to **Section 5.3.2 - Conversion Reaction** in the **HYSYS Simulation Basis** guide for details on creating Conversion Reaction Sets and Conversion Reactions.

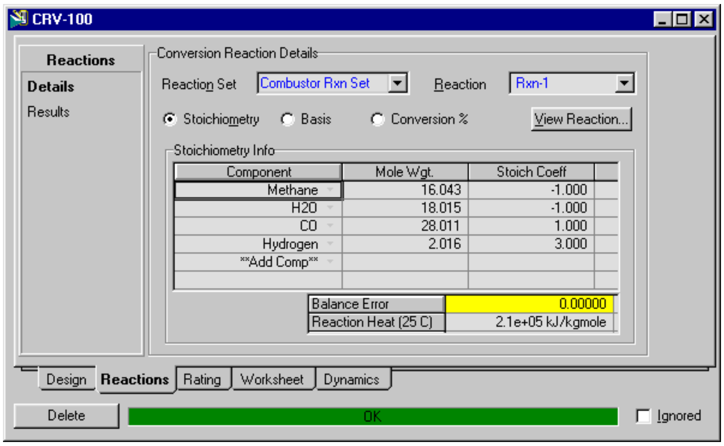
The Reactions tab, consists of the following pages:

- Details
- Results

Details Page

You can attach the reaction set to the operation and specify the conversion for each reaction in the set on the Details page. The reaction set can contain only conversion reactions.

Figure 8.4



The Details page consists of four objects as described in the table below.

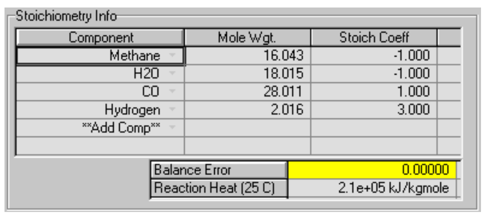
Object	Description
Reaction Set	Allows you to select the appropriate conversion reaction set.
Reaction	You must select the appropriate conversion reaction from the selected Reaction Set.

Object	Description
View Reaction button	Opens the Reaction property view for the reaction currently selected in the Reaction drop-down list. The Reaction property view allows you to edit the reaction.
[Radio buttons]	The three radio buttons on the Details page are: • Stoichiometry • Basis • Conversion The three radio buttons allow you to toggle between the Stoichiometry group, the Basis group or the Conversion group (each group is described in the following sections).

Stoichiometry Radio Button

When you select the Stoichiometry radio button, the Stoichiometry Info group appears. The Stoichiometry Info group allows you to examine the components involved in the selected reaction, their molecular weights as well as their stoichiometric coefficients.

Figure 8.5

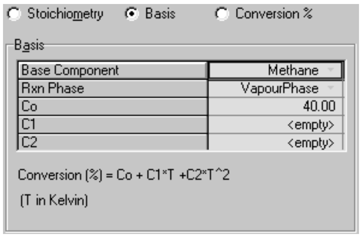


The Balance Error (for the reaction stoichiometry) and the Reaction Heat (Heat of Reaction at 25°C) are also shown for the current reaction.

Basis Radio Button

When you select the Basis radio button, the Basis group appears. In the Basis group, you can view the base component, the conversion, and the reaction phase for each reaction in the reaction set.

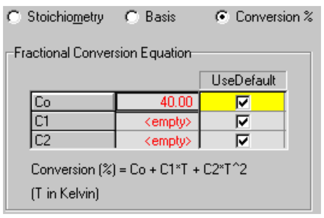
Figure 8.6



Conversion Radio Button

When you select the Conversion radio button, the Fractional Conversion Equation group appears. The Fractional Conversion Equation group allows you to implement a conversion model based on the *Conversion(%)* equation listed.

Figure 8.7



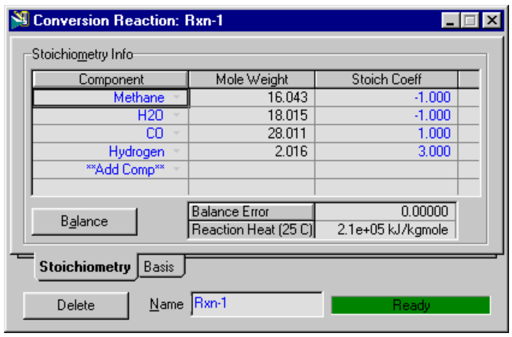
In the Fractional Conversion Equation group, parameters shown in red or blue colour indicate that the variable can be cloned.

The parameters for the attached conversion reaction(s) can be cloned as local variables belonging to the Conversion Reactor. Therefore, you can either use the parameters specified in the reaction(s) from the attached reaction set by clicking the Use Default checkbox or specifying locally the values within the Fractional Conversion Equation group.

View Reaction Button

When you click the **View Reaction** button, the Conversion Reaction property view of the reaction currently selected in the Reaction drop-down list appears.

Figure 8.8



Any changes made to the Conversion Reaction property view are made globally to the selected Reaction and any Reaction Sets which contain the Reaction. For example, if any change is made to the reaction shown in the figure above, the change is carried over to every other instance in which this Reaction is used. It is therefore recommended that changes which are Reactor specific (in other words, changes which are only meant to affect one Reactor) are made within the Reactions tab.

Results Page

The Results page displays the results of a converged reactor. The page consists of the Reactor Results Summary group which contains two radio buttons:

- Reaction Extents
- Reaction Balance

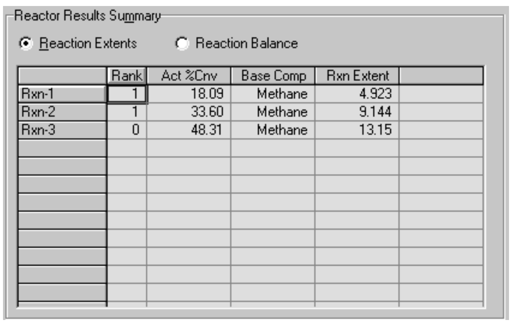
You can change the specified conversion for a reaction directly on this page.

The type of results displayed on the Results page depend on the radio button selected.

Reaction Extents Radio Button

When the Reaction Extents radio button is selected, the Results page appears as shown in the figure below.

Figure 8.9



The Reactor Results Summary group displays the following results for a converged reactor:

Result Field	Description
Rank	Displays the current rank of the reaction. For multiple reactions, lower ranked reactions occur first. When there are multiple reactions in a Reaction Set, HYSYS automatically ranks the reactions. A reaction with a lower ranking value occurs first. Each group of reactions of *equal rank* can have an overall specified conversion between 0% and 100%.
Actual % Conversion	Displays the percentage of the base component in the feed stream(s) which has been consumed in the reaction.
Base Component	The reactant to which the calculation conversion is based on.
Rxn Extent	Lists the molar rate consumption of the base component in the reaction divided by its stoichiometric coefficient appeared in the reaction.

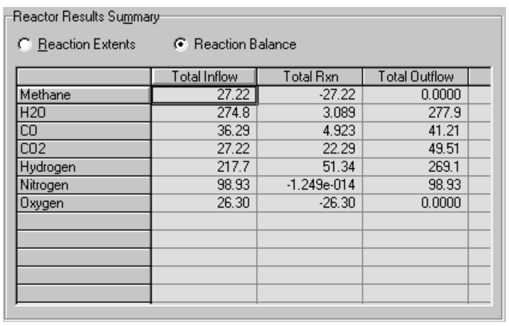
Notice that the actual conversion values do not match the specified conversion values. Rxn-3 proceeds first and is halted when a limiting reactant is exhausted. The sum of the specified conversions for Rxn-1 and Rxn-2 is 100%, so all of the remaining base component can be consumed, provided a limiting reactant is not fully consumed beforehand. All of the base component is consumed, and this is reflected in the actual conversion totalling 100%.

Any changes made to the global reaction affect all Reaction Sets to which the reaction is attached, provided local changes have not been made.

Reaction Balance Radio Button

When the Reaction Balance radio button is selected, the Reaction Balance option provides an overall component summary for the Conversion Reactor. All components which appear in the fluid package are shown here.

Figure 8.10

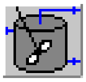


	Total Inflow	Total Rxn	Total Outflow
Methane	27.22	-27.22	0.0000
H2O	274.8	3.089	277.9
CO	36.29	4.923	41.21
CO2	27.22	22.29	49.51
Hydrogen	217.7	51.34	269.1
Nitrogen	98.93	-1.249e-014	98.93
Oxygen	26.30	-26.30	0.0000

Values appear after the solution of the reactor has converged. The Total Inflow rate, the Total Reacted rate and the Total Outflow rate for each component are provided on a molar basis. Negative values indicate the consumption of a reactant, while positive values indicate the appearance of a product.

8.2.3 CSTR Reactions Tab

For more information on Kinetic, Heterogeneous Catalytic and Simple Rate reactions, refer to **Chapter 5 - Reactions** in the **HYSYS Simulation Basis** guide.



CSTR icon

The CSTR is a vessel in which Kinetic, Heterogeneous Catalytic, and Simple Rate reactions can be performed. The conversion in the reactor depends on the rate expression of the reactions associated with the reaction type. The inlet stream is assumed to be perfectly (and instantaneously) mixed with the material already in the reactor, so that the outlet stream composition is identical to that of the reactor contents. Given the **reactor volume**, a **consistent rate expression** for each reaction and the **reaction stoichiometry**, the CSTR computes the conversion of each component entering the reactor.

On the Reactions tab, you can select a reaction set for the operation. You can also view the results of the solved reactor including the actual conversion of the base component. The actual conversion is calculated as the percentage of the base component that was consumed in the reaction.

$$X = \frac{N_{A_{in}} - N_{A_{out}}}{N_{A_{in}}} \times 100\% \tag{8.4}$$

where:

X = actual % conversion

N_{Ain} *= base component flowrate into the reactor*

N_{Aout} *= base component flowrate (same basis as the inlet rate) out of the reactor*

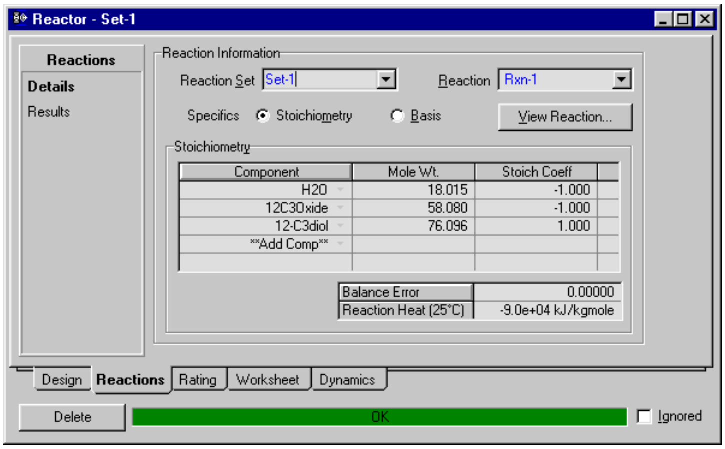
The Reactions tab contains the following pages:

- Details
- Results

Details Page

The Details page allows you to attach the appropriate reaction set to the operation.

Figure 8.11



As mentioned earlier in this section, the selected reaction set can contain only Kinetic, Heterogeneous Catalytic, and Simple Rate reactions.

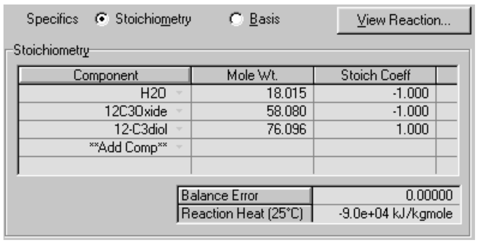
The page consists of four objects, which are described in the table below.

Object	Description
Reaction Set	Allows you to select the reaction set you want to use in the reactor.
Reaction	Allows you to select the reaction you want to use in the reactor.
View Reaction	Opens the Reaction property view for the selected Reaction. This allows you to edit the reaction globally.
Specifics	Toggles between the Stoichiometry group or the Basis group (the groups are described in the following sections).

Stoichiometry Radio Button

When you select the Stoichiometry radio button, the Stoichiometry group appears. The Stoichiometry group allows you to examine the components involved in the currently selected reaction, their molecular weights as well as their stoichiometric coefficients.

Figure 8.12

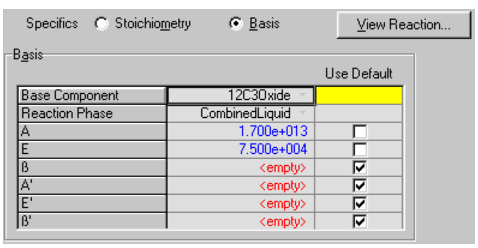


The Balance Error (for the reaction stoichiometry) and the Reaction Heat (Heat of Reaction at 25°C) are also shown for the current reaction.

Basis Radio Button

When you select the Basis radio button, the Basis group appears. In the Basis group, you can view the base component, the reaction rate parameters (for example A, E, ß, A', E', and ß') and the reaction phase for each reaction in the attached set.

Figure 8.13



You can view the properties for a specific reaction by selecting the reaction from the Reaction drop-down list, and its data appears in the Basis group.

Changes can be made to the reaction rate parameters (frequency factor, A, activation energy, E, and ß), but these changes are reflected only in the active reactor. The changes do not affect the global reaction.

To return the global reaction values, select the appropriate Use Default checkbox. For instance, if you have made a change to the forward reaction activation energy (E), the Use Default E checkbox is inactive. Select this checkbox to return to the global E value.

Results Page

The Results page displays the results of a converged reactor. The page is made up of the Reaction Results Summary group which contains two radio buttons:

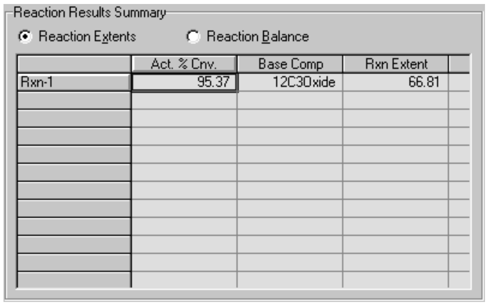
- Reaction Extents
- Reaction Balance

Reaction Extents Radio Button

When you select the Reaction Extents radio button, the Reaction Extents option displays the following results for a converged reactor:

Result Field	Description
Actual % Conversion	Displays the percentage of the base component in the feed stream(s) which has been consumed in the reaction.
Base Component	The reactant to which the conversion is applied.
Rxn Extent	Lists the molar rate consumption of the base component in the reaction divided by its stoichiometirc coefficient appeared in the reaction.

Figure 8.14

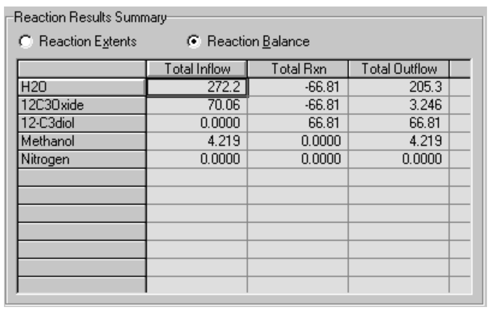


	Act. % Cnv.	Base Comp	Rxn Extent
Rxn-1	95.37	12C3Oxide	66.81

Reaction Balance Radio Button

When you select the Reaction Balance radio button, the Reaction Balance option provides an overall component summary for the CSTR. All components which appear in the fluid package are shown here.

Figure 8.15

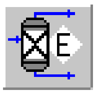


	Total Inflow	Total Rxn	Total Outflow
H2O	272.2	-66.81	205.3
12C3Oxide	70.06	-66.81	3.246
12-C3diol	0.0000	66.81	66.81
Methanol	4.219	0.0000	4.219
Nitrogen	0.0000	0.0000	0.0000

Values appear after the solution of the reactor has converged. The Total Inflow rate, the Total Reacted rate and the Total Outflow rate for each component are provided on a molar basis. Negative values indicate the consumption of a reactant, while positive values indicate the appearance of a product.

8.2.4 Equilibrium Reactor Reactions Tab

Refer to **Section 5.3.3 - Equilibrium Reaction** in the **HYSYS Simulation Basis** guide for details on creating and installing Equilibrium Reactions.



Equilibrium Reactor icon

The Equilibrium reactor is a vessel which models equilibrium reactions. The outlet streams of the reactor are in a state of chemical and physical equilibrium. The reaction set which you attach to the Equilibrium Reactor can contain an unlimited number of equilibrium reactions, which are simultaneously or sequentially solved. Neither the components nor the mixing process need be ideal, since HYSYS can compute the chemical activity of each component in the mixture based on mixture and pure component fugacities.

You can also examine the actual conversion, the base component, the equilibrium constant, and the reaction extent for each reaction in the selected reaction set. The conversion, the equilibrium constant and the extent are all calculated based on the equilibrium reaction information which you provided when the reaction set was created.

Any changes made to the global reaction affect all reaction sets to which the reaction is attached, provided local changes have not been made.

The Reactions tab contains the following pages:

- Details
- Results

Details Page

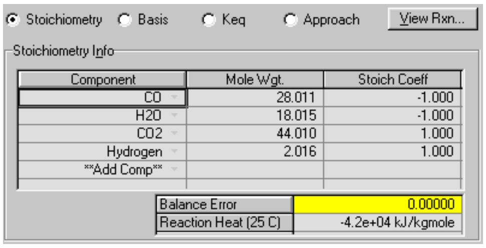
The Details page consists primarily of four radio buttons:

- Stoichiometry
- Basis
- Ln[K]
- Table

Stoichiometry Radio Button

When you select the Stoichiometry radio button, the Stoichiometry Info group appears. The Stoichiometry group allows you to view the stoichiometric formula of the reaction currently selected in the Reaction drop-down list.

Figure 8.16



Changes made to the global reaction affect all reaction sets which contain the reaction, and thus all operations to which the reaction set is attached.

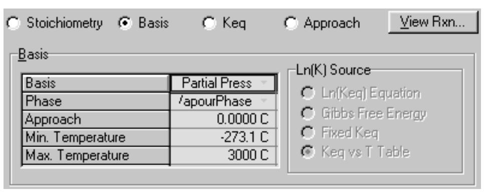
The Balance Error (for the reaction stoichiometry) and the Reaction Heat (Heat of Reaction at 25°C) are also shown for the current reaction.

Basis Radio Button

Refer to **Section 5.3.3 - Equilibrium Reaction** of the **HYSYS Simulation Basis** guide for details on Equilibrium Constant source.

When you select the Basis radio button, Basis group appears.

Figure 8.17



The Basis group allows you to view or edit (locally) various information for each reaction in the reaction set including the:

- Basis for the equilibrium calculations.
- Phase in which the reaction occurs.
- Temperature Approach of the equilibrium composition.

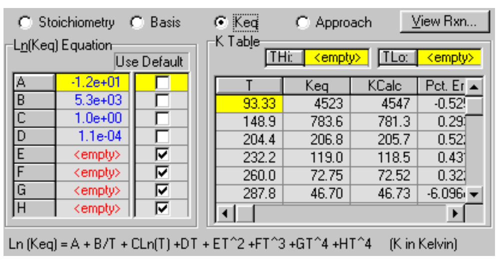
The temperature range for the equilibrium constant, and the source for the calculation of the equilibrium constant is also shown.

Keq Radio Button

Refer to **Section 5.3.3 - Equilibrium Reaction** of the **HYSYS Simulation Basis** guide for details on the Equilibrium Constant source.

When you select the Keq radio button, the Ln(keq) group and K Table appears.

Figure 8.18



Refer to the section on the **Basis Radio Button** for more information.

The Ln(keq) group displays the Ln(Keq) relationship which may vary depending upon the Ln(K) Source value selected for the reaction.

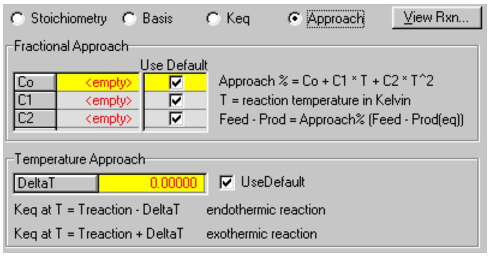
When you select the Ln(Keq) Equation radio button in the Ln(K) Source group, the parameters of the equilibrium constant equation appear. These values are either specified when the reaction was created or are calculated by HYSYS. If a fixed equilibrium constant was provided, it is shown here.

Any of the parameters in the Ln(K) Equation group can be modified on this page. Changes made to the parameters only affect the selected reaction in the current reactor. After a change has been made, you can have HYSYS return the original calculated value by selecting the appropriate **Use Default** checkbox.

Approach Radio Button

When you select the Approach radio button, the Fractional Approach group and the Temperature Approach group appear.

Figure 8.19



For each reaction in the reaction set, a fractional approach equation as a function of temperature is provided. Any of the parameters in the Approach % equation can be modified on this page. Changes made to the parameters only affect the selected reaction in the current reactor. After a change has been made, you can have HYSYS return the original calculated value by selecting the appropriate **Use Default** checkbox.

For more detailed information on equilibrium reactions, refer to **Chapter 5 - Reactions** in the **HYSYS Simulation Basis** guide.

You can edit a reaction by clicking the **View Reaction** button. The property view for the highlighted reaction appears.

You can change the specified conversion for a reaction directly on this page.

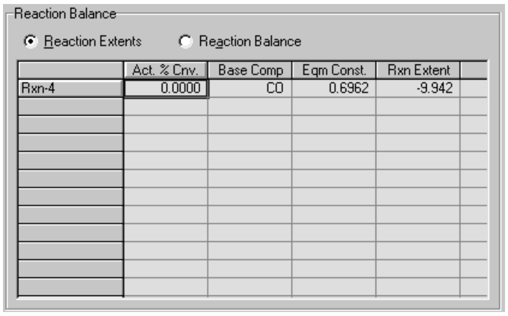
Results Page

The Results page displays the results of a converged reactor. The page is made up of the Results Summary group which contains two radio buttons:

- Reaction Extents
- Reaction Balance

Reaction Extents

Figure 8.20



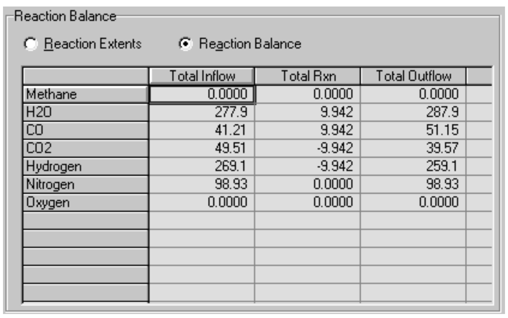
When you select the Reaction Extents radio button, the option displays the following results for a converged reactor:

Result Field	Description
Actual % Conversion	Displays the percentage of base component in the feed stream(s) which has been consumed in the reaction. The actual conversion is calculated as the percentage of the base component that was consumed in the reaction. $$X = \frac{N_{A_{in}} - N_{A_{out}}}{N_{A_{in}}} \times 100\% \qquad (8.5)$$ *where:* *X = actual % conversion* N_{Ain} *= base component flowrate into the reactor* N_{Aout} *= base component flowrate (same basis as the inlet rate) out of the reactor*
Base Component	The reactant to which the conversion is applied.

Result Field	Description
Eqm Const.	The equilibrium constant is calculated at the reactor temperature by the following: $$\ln K = A + \frac{B}{T} + C\ln T + DT \quad (8.6)$$ *where:* *T = reactor temperature, K* *A, B, C, D = equation parameters* The four parameters in **Equation (8.6)** are calculated by HYSYS if they are not specified during the installation of the equilibrium reaction. The four parameters for each equilibrium equation are listed on the **Rxn Ln(K)** page.
Rxn Extent	Lists the molar rate consumption of the base component in the reaction divided by its stoichiometirc coefficient appeared in the reaction.

Reaction Balance

Figure 8.21



	Total Inflow	Total Rxn	Total Outflow
Methane	0.0000	0.0000	0.0000
H2O	277.9	9.942	287.9
CO	41.21	9.942	51.15
CO2	49.51	-9.942	39.57
Hydrogen	269.1	-9.942	259.1
Nitrogen	98.93	0.0000	98.93
Oxygen	0.0000	0.0000	0.0000

When you select the Reaction Balance radio button, the Reaction Balance option provides an overall component summary for the Equilibrium Reactor. All components which appear in the component list related to the fluid package are shown here.

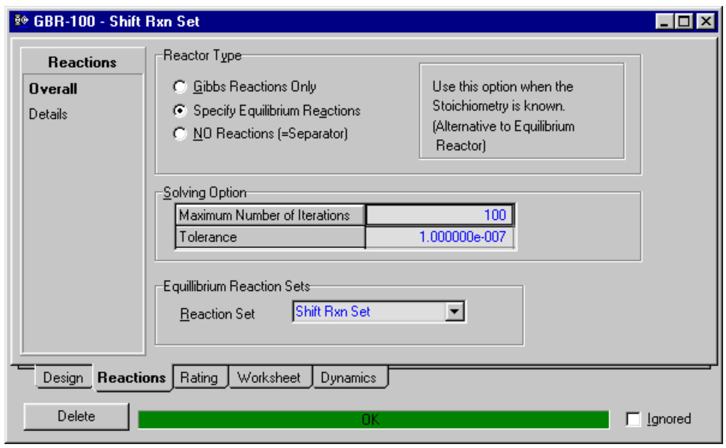
Values appear after the solution of reactor has converged. The Total Inflow rate, the Total Reacted rate, and the Total Outflow rate for each component are provided on a molar basis. Negative values indicate the consumption of a reactant, while positive values indicate the appearance of a product.

8.2.5 Gibbs Reactor Reactions Tab

Gibbs Reactor icon

The Gibbs Reactor calculates the exiting compositions such that the phase and chemical equilibria of the outlet streams are attained. However, the Gibbs Reactor does not need to make use of a specified reaction stoichiometry to compute the outlet stream composition. The condition that the Gibbs free energy of the reacting system is at a minimum at equilibrium is used to calculate the product mixture composition. As with the Equilibrium Reactor, neither pure components nor the reaction mixture are assumed to behave ideally.

Figure 8.22



The versatility of the Gibbs Reactor allows it to function solely as a separator, as a reactor which minimizes the Gibbs free energy without an attached reaction set or as a reactor which accepts equilibrium reactions. When a reaction set is attached, the stoichiometry involved in the reactions is used in the Gibbs Reactor calculations.

The Reactions tab contains the following pages:

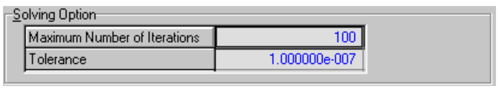
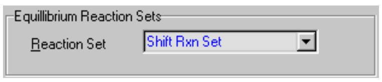
- Overall
- Details

Overall Page

You must first select the reactor type on the Overall page. The objects that appear depend on the radio button you selected in the Reactor Type group. You can then attach a reaction set if necessary, and you can specify the vessel parameters on the Rating tab.

Reactor Type Group

In the Reactor Type group, select the radio button to define the method which HYSYS uses to solve the Gibbs Reactor. The table below describes the radio buttons.

Radio Button	Description
Gibbs Reactions Only	No reaction set is required as HYSYS solves the system by minimizing the Gibbs free energy while attaining phase and chemical equilibrium. You can also customize the maximum iteration number and equilibrium error tolerance in the Solving Option group. 
Specify Equilibrium Reactions	Displays the Equilibrium Reaction Sets group. When a reaction set is attached, the Gibbs Reactor is solved using the stoichiometry of the reactions involved. The Gibbs minimization function uses the extents of the attached reactions while setting any unknowns to zero. 
NO Reactions (=Separator)	The Gibbs Reactor is solved as a separator operation, concerned only with phase equilibrium in the outlet streams.

Details Page

The Details page consists of one group, the Gibbs Reaction Details group. The group consists of two radio buttons:

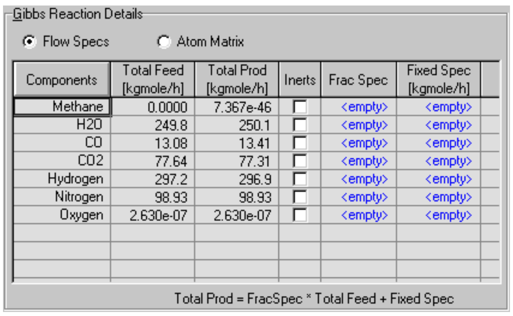
- Flow Specs
- Atom Matrix

The information that is viewable on the page depends on which of the two radio buttons is selected.

Flow Specs Option

When you select the Flow Specs radio button, a property view similar to the one in the figure below appears.

Figure 8.23



Components	Total Feed [kgmole/h]	Total Prod [kgmole/h]	Inerts	Frac Spec	Fixed Spec [kgmole/h]
Methane	0.0000	7.367e-46		<empty>	<empty>
H2O	249.8	250.1		<empty>	<empty>
CO	13.08	13.41		<empty>	<empty>
CO2	77.64	77.31		<empty>	<empty>
Hydrogen	297.2	296.9		<empty>	<empty>
Nitrogen	98.93	98.93		<empty>	<empty>
Oxygen	2.630e-07	2.630e-07		<empty>	<empty>

You can view the component feed and product flowrates on a molar basis. You can also designate any of the components as inert or specify a rate of production for a component.

Inert species are excluded from the Gibbs free energy minimization calculations. When the **Inerts** checkbox is selected for a component, values of 1 and 0 appear respectively in the associated Frac Spec and Fixed Spec cells, which indicates that the component feed flowrate equals the product flowrate.

You may want to specify the rate of production of any component in your reactor as a constraint on the equilibrium composition. The component product flowrate is calculated as follows, based on your input of a Frac Spec value and a Fixed Spec value:

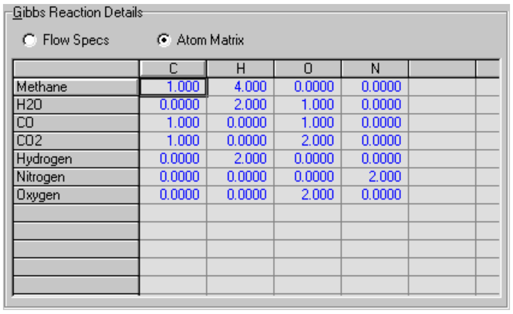
$$\text{Total Prod} = \text{FracSpec} \times \text{Total Feed} + \text{FixedSpec} \tag{8.7}$$

The Gibbs Reactor attempts to meet that flowrate in calculating the composition of the outlet stream. If the constraint cannot be met, a message appears alerting you to that effect.

Atom Matrix Option

When you select the Atom Matrix radio button, you can specify the atomic composition of any species for which the formula is unknown or unrecognized.

Figure 8.24



	C	H	O	N
Methane	1.000	4.000	0.0000	0.0000
H2O	0.0000	2.000	1.000	0.0000
CO	1.000	0.0000	1.000	0.0000
CO2	1.000	0.0000	2.000	0.0000
Hydrogen	0.0000	2.000	0.0000	0.0000
Nitrogen	0.0000	0.0000	0.0000	2.000
Oxygen	0.0000	0.0000	2.000	0.0000

The atomic matrix input form displays all components in the case with their atomic composition as understood by HYSYS. You have the option to enter the composition of an unrecognized compound or to correct the atomic composition of any compound.

8.2.6 Rating Tab

For information on specifying information on the Sizing Page, refer to the **HYSYS Dynamic Modeling** guide.

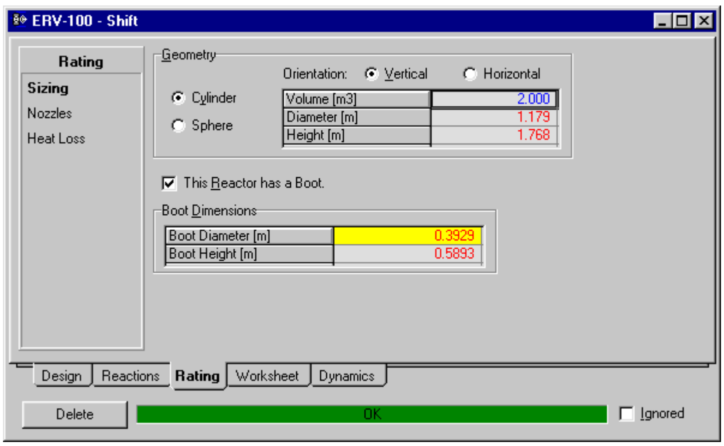
The Rating tab includes the Sizing, Nozzles, and Heat Loss pages. Although most of the information on the three pages is not relevant when working in the Steady State mode, sizing a reactor plays an important role in calculating the holdup time.

You are required to specify the rating information only when working with a dynamics simulation.

Sizing Page

You can define the geometry of the unit operation on the Sizing page. Also, you can indicate whether or not the unit operation has a boot associated with it. If it does, then you can specify the boot dimensions.

Figure 8.25



The page consists of three main objects, which are described in the table below.

Object	Description
Geometry	Allows you to specify the vessel geometry.
This Reactor has a Boot	When activated, the Boot Dimensions group appears.
Boot Dimensions	Allows you to specify the boot dimensions of the vessel.

Geometry Group

The Geometry group contains five objects which are described in the table below.

Object	Description
Cylinder / Sphere	Toggles the shape of the vessel between Sphere and Cylinder. This affects the number of specifications required as well as the method of volume calculation. If you select the Cylinder, and you have specified the diameter and height; the vessel volume is calculated as: $$V_{reactor} = \left(\frac{Diameter^2}{4}\pi \times Height\right) + V_{boot} \quad (8.8)$$ If you select Sphere, and you have specified either the height or diameter; the vessel volume is calculated as: $$V_{reactor} = \frac{(Height\ or\ Diameter)^3\pi}{6} + V_{boot} \quad (8.9)$$ *where:* *$V_{reactor}$ = volume of the reactor* *V_{boot} = volume of the boot* *Height, Diameter = values taken from the respective fields*
Orientation	Allows you to select the orientation of the vessel. There are two options: • **Horizontal**. The ends of the vessels are horizontally orientated. • **Vertical**. The ends of the vessel are vertically orientated.

Object	Description
Volume	Contains the total volume of the vessel. There are three possibilities for values in this field: • If the height and/or diameter have been entered, this field displays the value calculated using either **Equation (8.8)** or **Equation (8.9)**. • If you enter a value into this field and either the height (length) or diameter is specified, HYSYS back calculates the other parameter using either **Equation (8.8)** or **Equation (8.9)**. This is **only** possible with cylindrical vessels as spherical vessels have the height equal to the diameter. • If you enter a value into this field (and only this field) both the height (length) and diameter are calculated assuming a ratio of 3/2 (in other words, Height:Diameter ratio).
Diameter	Holds the diameter of the vessel. If the vessel is a Sphere, then it is the same value as the Height (Length).
Height / Length	Holds the height or length of the vessel depending on the vessels orientation (vertical or horizontal). If the vessel is a Sphere, then it is the same value as the diameter.

The Geometry group contains three fields:

- Volume
- Diameter
- Height (or Length depending on orientation)

If you specify the Volume then you are not required to specify the other two parameters as HYSYS calculates a Height (or Length) and Diameter assuming a ratio of Height to Diameter of 3/2.

You can change the default ratio, by specifying one of the two dimensions (either Height or Diameter) and the third is automatically calculated using either Equation (8.8) or Equation (8.9).

Boot Dimensions

If the reactor you are rating has a boot, you can include its volume in the total vessel volume by selecting the **This Reactor has a Boot** checkbox. The Boot Dimensions group appears.

The Boot Dimensions group consists of two fields, which are described in the table below.

Field	Description
Boot Diameter	The diameter of the boot. The default value is usually 1/3 the reactor diameter.
Boot Height	The height of the boot which is defaulted at 1/3 the reactor diameter (sphere) or 1/3 the reactor height or length (cylinder).

The volume of the boot is calculated using a simple cylindrical volume calculation:

$$V_{Boot} = \pi\left(\frac{Boot\ Diameter}{2}\right)^2 \times (Boot\ Height\ or\ Boot\ Length) \tag{8.10}$$

and the default boot volume is:

$$\begin{aligned} V_{Boot} &= \pi\left(\frac{Diameter}{6}\right)^2 \times \frac{Diameter}{3} \\ &= \frac{\pi(Diameter)^3}{72} \end{aligned} \tag{8.11}$$

The total Reactor volume can estimated using the boot diameter, boot height or the default boot volume.

Nozzles Page

Refer to **Section 1.3.6 - Nozzles Page** for more information.

The Nozzles page contains information regarding the elevation and diameter of the nozzles.

Unlike steady state vessel operations, the placement of feed and product nozzles on a dynamic reactor operation has physical meaning. The composition of the exit stream depends on the exit stream nozzle’s location and diameter in relation to the physical holdup level in the vessel.

- If the product nozzle is located below the liquid level in the vessel, the exit stream draws material from the liquid holdup.

- If the product nozzle is located above the liquid level, the exit stream draws material from the vapour holdup.
- If the liquid level lies across a nozzle, the phase fraction of liquid in the product stream varies linearly with how far up the liquid is in the nozzle.

Essentially, all vessel operations in HYSYS are treated similarly. The composition and phase fractions (in other words, fraction of each phase) of every product stream depends solely on the relative levels of each phase in the holdup and the location the product nozzles.

A vapour product nozzle does not necessarily produce pure vapour and a 3-phase separator may not produce two distinct liquid phase products from its product nozzles.

Heat Loss Page

For information refer to **Heat Loss Page** section in **Chapter 10 - Separation Operations**.

The Heat Loss page allows you to specify which Heat Loss Model you want to implement, and to define the parameters associated with each model.

8.2.7 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

The PF Specs page is relevant to dynamics cases only.

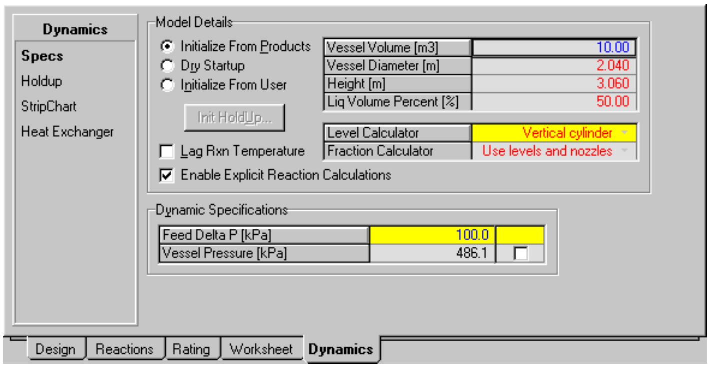
8.2.8 Dynamics Tab

If you are working exclusively in Steady State mode, you are not required to change any information on the pages accessible through this tab.

Specs Page

The Specs page contains information regarding initialization modes, vessel geometry, and vessel dynamic specifications.

Figure 8.26



Model Details

You can determine the composition and amount of each phase in the vessel holdup by specifying different initialization modes. HYSYS forces the simulation case to re-initialize whenever the initialization mode is changed.

The radio buttons in the Model Details group are described in the table below.

Initialization Mode	Description
Initialize from Products	The composition of the holdup is calculated from a weighted average of all products exiting the holdup. A PT flash is performed to determine other holdup conditions. The liquid level is set to the value indicated in the Liq Volume Percent field.
Dry Startup	The composition of the holdup is calculated from a weighted average of all feeds entering the holdup. A PT flash is performed to determine other holdup conditions. The liquid level in the Liq Volume Percent field is set to zero.
Initialize from User	The composition of the liquid holdup in the vessel is user specified. The molar composition of the liquid holdup can be specified by clicking the Init Holdup button. The liquid level is set to the value indicated in the Liq Volume Percent field.

The Enable Explicit Reaction Calculations is defaulted to be used for dynamic run reaction solver.

The Lag Rxn Temperature is designed to speed up the dynamic run for the reaction solver when the run has to invoke the steady state reaction solver. Mathematically, when you select the **Lag Rxn Temperature** checkbox, the reaction solver flashes with the explicit Euler method. Otherwise, for a dynamic run, the steady state reaction solver always flashes with the implicit Euler methods which could be slow with many iterations.

The Lag Rxn Temperature may cause some instability due to the nature of the explicit Euler method. But it must compromise with the dynamic step size.

In the Model Details group, you can specify the vessel geometry parameters.

- Vessel Volume
- Vessel Diameter
- Vessel Height (Length)
- Vessel Geometry (Level Calculator)

The vessel geometry parameters can be specified in the same manner as those specified in the Geometry group for the Sizing page of the Rating tab.

Liquid Volume Percent

You can modify the level in the vessel at any time. HYSYS then uses that level as an initial value when the Integrator has started, depending on the initialization mode you selected.

Fraction Calculator

The Fraction Calculator determines how the level in the tank, and the elevation and diameter of the nozzle affects the product composition.

The Fraction Calculator defaults to the correct mode for all unit operations and does not typically require any changing.

The following is a description of the Fraction Calculator option:

Use Levels and Nozzles.The nozzle location and vessel liquid level affect the product composition as detailed in the **Nozzles Page** of **Section 8.2.6 - Rating Tab**.

Dynamic Specifications

The frictional pressure loss at the feed nozzle is a dynamic specification in HYSYS. It can be specified in the Feed Delta P field. The frictional pressure losses at each product nozzle are automatically set to zero by HYSYS.

It is recommended that you enter a value of zero in the Feed Delta P field because a fixed pressure drop in the vessel is not realistic for all flows.

If you want to model friction loss at the inlet and exit stream, it is suggested you add valve operations. In this case, flow into and out of the vessel is realistically modeled.

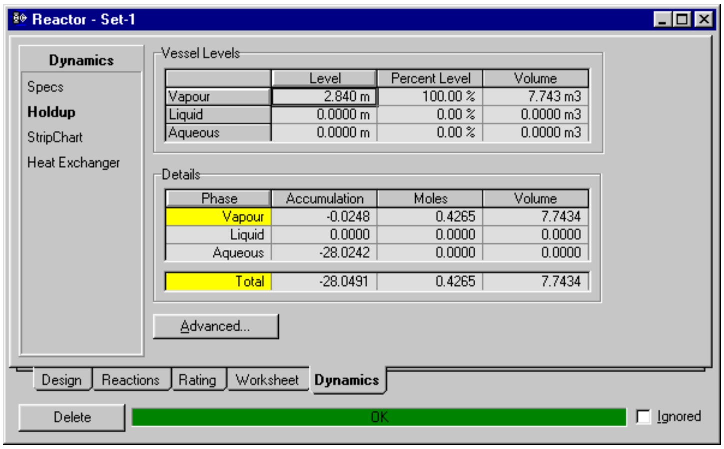
The vessel pressure can also be specified. This specification can be made active by selecting the checkbox beside the **Vessel Pressure** field. This specification is typically not set since the pressure of the vessel is usually a variable and determined from the surrounding pieces of equipment.

Holdup Page

Refer to **Section 1.3.3 - Holdup Page** for more information.

The Holdup page contains information regarding the properties, composition, and amount of the holdup.

Figure 8.27



The Vessel Levels group displays the following variables for each of the phases available in the vessel:

- Level. Height location of the phase in the vessel.
- Percent Level. Percentage value location of the phase in the vessel.
- Volume. Amount of space occupied by the phase in the vessel.

Stripchart Page

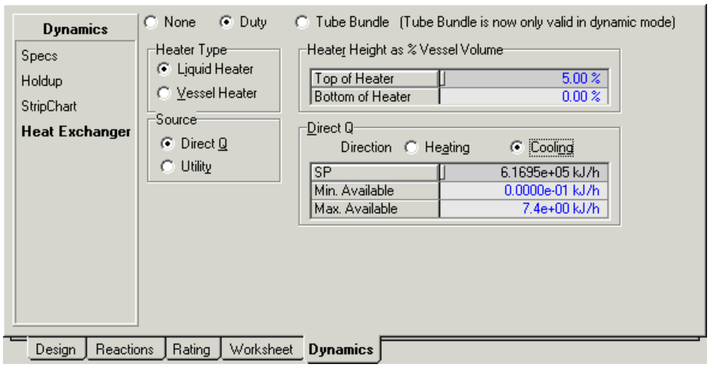
Refer to **Section 1.3.7 - Stripchart Page/Tab** for more information.

The Stripchart page allows you to select and create default strip charts containing various variable associated to the operation.

Heat Exchanger Page

The Heat Exchanger page allows you to select whether the reactor is heated, cooled, or left alone. You can also select the method used to heat or cool the reactor.

Figure 8.28



The options available in the **Heat Exchanger** page depends on which radio button you select:

Refer to **Duty Radio Button** for more information.

- If you select the **None** radio button, this page is blank and you do not have to specify an energy stream in the **Connections** page (from the **Design** tab) for the reactor operation to solve.
- If you select the **Duty** radio button, this page contains the standard heater or cooler parameters and you have to specify an energy stream in the **Connections** page (from the **Design** tab) for the reactor operation to solve.
- The **Tube Bundle** radio button option is not available for the reactor operations.

If you switch from Duty option to None option, HYSYS automatically disconnects the energy stream associated to the Duty options.

8.3 Yield Shift Reactor

The Yield Shift reactor unit operation supports efficient modeling of reactors by using data tables to perform shift calculations. The operation can be used for complex reactors where no model is available, or where models that are too computationally expensive.

Theory

There are two methods to configure the reaction in the Yield Shift reactor: Yield Only or Percent Conversion. Depending on what information you supply the reactor automatically use the appropriate equation to solve the reaction.

Product Stream Mass Fractions

The following equations are used to calculate the product stream mass fractions:

- For Component Percent Conversion method:

$$y_k = x_k \times (1 - \mathrm{conv}_k) + \mathrm{cur\_yield}_k \times \mathrm{conv}_{total} \tag{8.12}$$

where:

y_k = *mass fraction of component k in product stream*

The sum of the component mass fraction must equal one.

x_k = *mass fraction of component k in feed stream*

conv_k = *component type (reacting or non-reacting) for component k*

$$\mathrm{conv}_{total} = \sum_{k=0}^{NC} x_k \times \mathrm{conv}_k \tag{8.13}$$

$$\mathrm{cur\_yield}_k = \mathrm{base\_yield}_k + \mathrm{total\_shift}_k \tag{8.14}$$

NC = number of components

$\mathrm{base\_yield}_k$ = *base yield of component k*

There are two methods to obtain the base yield value:

- Calculate the value from raw data, see **Design Data:Base Page**.
- Specify the value, see **Base Yields Page**.

$$\mathrm{total\_shift}_k = \sum_{i=0}^{NV} \sum_{j=0}^{NR^i} [(\mathrm{cur\_adj}_i^j - \mathrm{base\_adj}_i^j) \times (\mathrm{base\_shift}_i^j)_k \times \mathrm{eff}_i] \tag{8.15}$$

$$\mathrm{cur\_adj}_i^j = Max[p_i^{j,min}, Min(p_i^{j,max}, \mathrm{cur\_value}_i)] \tag{8.16}$$

$$\mathrm{base\_adj}_i^j = Max[p_i^{j,min}, Min(p_i^{j,max}, \mathrm{base\_value}_i)] \tag{8.17}$$

NV = number of input variables

NR^i = *number of ranges for each input variable i*

$(\mathrm{base\_shift}_i^j)_k$ = *base shift value for component k*

The sum of the base shift values for all the components should equal zero.

eff_i = *efficiency for design variable i, the values are user-specified in the* ***Efficiencies Page***

$p_i^{j,min}$ = *minimum range value of dataset j of design variable i*

$p_i^{j,max}$ = *maximum range value of dataset j of design variable i*

If the range values are not specified, HYSYS assumes negative and positive infinity values for minimum and maximum range respectively.

cur_value_i = *current value for design variable i*

base_value_i = *base value for design variable i*

- For Yield Only Conversion method:

$$y_k = x_k + \text{cur\_yield}_k \times \text{conversion}_{total} \tag{8.18}$$

where:

y_k = *mass fraction of component k in product stream*

The sum of the component mass fraction must equal one.

x_k = *mass fraction of component k in feed stream*

$\text{conversion}_{total}$ = *total conversion value for the reaction*

$$\text{cur\_yield}_k = \text{base\_yield}_k + \text{total\_shift}_k \tag{8.19}$$

NC = number of components

base_yield_k = *base yield of component k*

There are two methods to obtain the base yield value:

- Calculate the value from raw data, see **Design Data:Base Page**.
- Specify the value, see **Base Yields Page**.

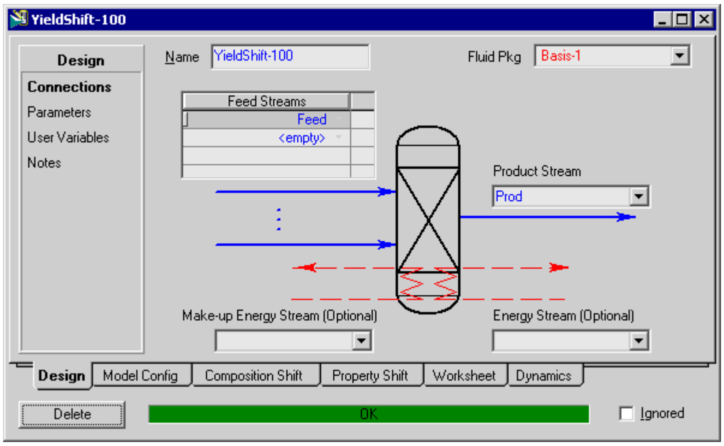
total_shift_k = *total shift value, see* ***Equation (8.15)***

8.3.1 Yield Shift Reactor Property View

The Yield Shift Reactor property view contains the following tabs:

- Design
- Model Config
- Composition Shift
- Property Shift
- Worksheet
- Dynamics

Figure 8.29



8.3.2 Design Tab

The Design tab contains the options that enables you to configure the Yield Shift reactor. The options are grouped in the following pages:

- Connections
- Parameters
- User Variables

  The User Variables page enables you to create and implement your own user variables for the current operation.

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

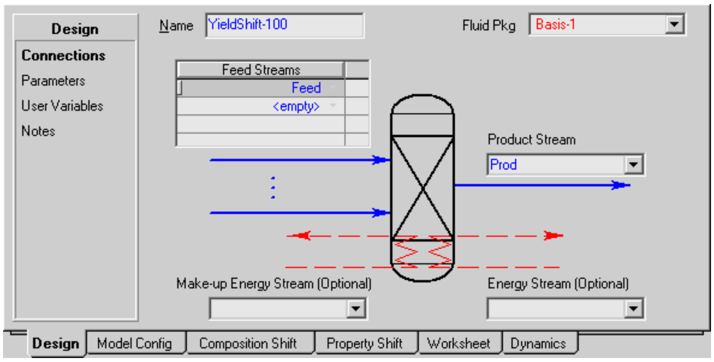
For more information on the Notes page, refer to **Section 1.3.5 - Notes Page/Tab**.

- Notes
  The Notes page enables you to add relevant comments which are exclusively associated with the unit operation.

Connections Page

The Connections page enables you to configure the material and energy streams flowing in and out of the reactor.

Figure 8.30

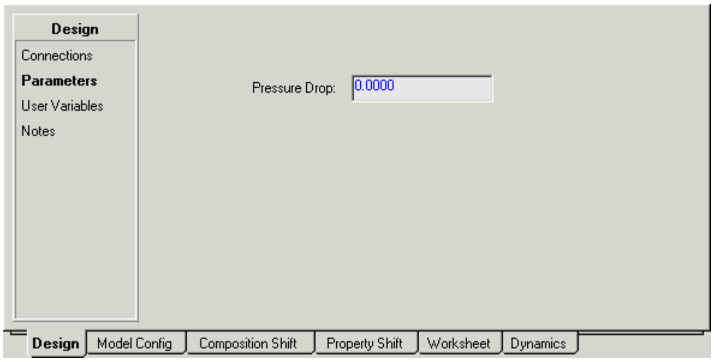


Object	Description
Name field	Enables you to modify the name of the reactor.
Fluid Pkg field	Enables you to select the fluid package associated to the reactor.
Feed Streams table	Enables you to specify or select inlet streams flowing into the reactor.
Product Stream field	Enables you to specify or select an outlet stream flowing out of the reactor.
Make-up Energy Stream (Optional)	Enables you to connect or create a make-up energy stream if one is required for the operation. If you specify any heat adjustment for the reaction, HYSYS automatically creates a make-up energy stream to represent the heat transfer.
Energy (Optional)	Enables you to connect or create an energy stream if one is required for the operation.

Parameters Page

The Parameters page allows you to specify the pressure drop of the reactor.

Figure 8.31



The **Pressure Drop** field enables you to specify the pressure drop in the vessel of the reactor. The pressure drop is defined as:

$$\Delta P = P_{feed} - P_{product} \tag{8.20}$$

where:

ΔP = *pressure drop in vessel (Delta P)*

$P_{product}$ = *pressure of the product stream*

P_{feed} = *pressure of the feed stream, assumed to be the lowest pressure of all the feed streams*

The default pressure drop across the vessel is zero.

8.3.3 Model Config Tab

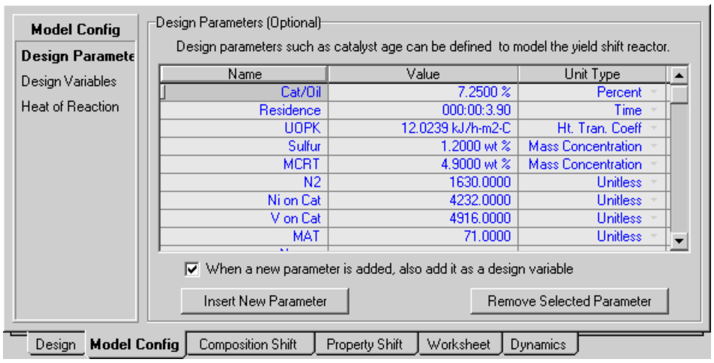
The Model Config tab contains the options for configuring the reactor. These options are split into the following pages:

- Design Parameters
- Design Variables
- Heat of Reaction

Design Parameters Page

The Design Parameters page enables you to specify other design parameters that affect the reaction in the reactor.

Figure 8.32



These parameters are optional and you do not have to supply any parameter information to get the reactor to solve.

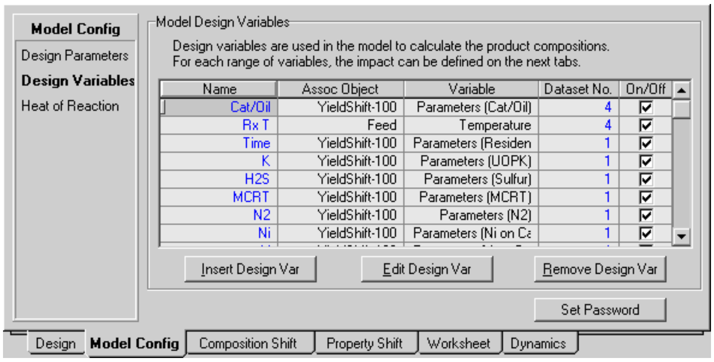
Object	Description
Name column	Enables you to change the name of the selected design parameter.
Value column	Enables you to specify the design parameter value.
Unit Type column	Enables you to select the unit type for the design parameter.
When a new parameter... checkbox	Enables you to toggle between adding or not adding a new design variable every time a new design parameter is added.

Object	Description
Insert New Parameter button	Enables you to add a new design parameter.
Remove Selected Parameter button	Enables you to remove the selected operation parameter in the Operating Parameters table. You can select multiple parameters by pressing and holding the **CTRL** or **SHIFT** key while selecting the parameters.

Design Variables Page

The Design Variable page enables you to insert, edit, and remove variables used in the reaction calculations for the reactor.

Figure 8.33



Object	Description
Name column	Enables you to modify the name of the design variable.
Assoc Object column	Displays the name of the object associated to the design variable.
Variable column	Displays the design variable type.
Dataset No column	Enables you to specify the number of data set/value is available for the design variable.
On/Off checkbox	Enables you to toggle between acknowledging or ignoring the design variable during calculation. A clear checkbox indicates you are ignoring the design variable.
Insert Design Var button	Enables you to add a design variable using the property view similar to the Variable Navigator property view.

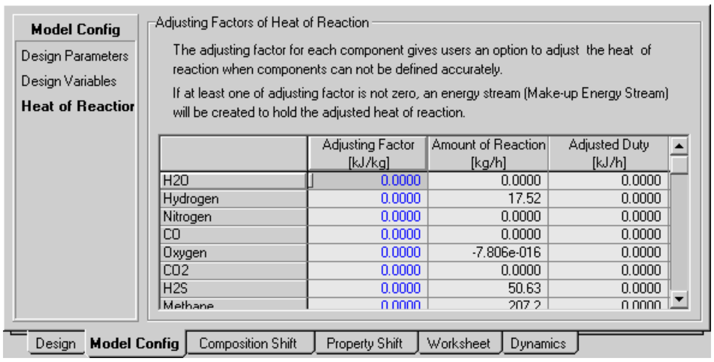
Refer to **Section 1.3.9 - Variable Navigator Property View** for more information.

Object	Description
Edit Design Var button	Enables you to change the selected design variable to a different variable.
Remove Design Var button	Enables you to remove the selected design variable from the reactor.
Set Password button	Enables you to set the password for the reactor. This button is only available if you have not set a password for the reactor. The password feature is available for users to protect the configuration data of the Yield Shift reactor. The password feature protects the proprietary property of the Yield Shift reactor configuration, while enables the reactor to be shared among other HYSYS users.
Change Password button	Enables you to change the password for the reactor. This button is only available if the reactor already contains a password.

Heat of Reaction Page

The Heat of Reaction page enables you to specify heat transfer that occurs during reaction.

Figure 8.34



Object	Description
Adjusting Factor column	Enables you to specify the heat transfer that occur for the associate component in the reactor. If you specify a value (other than zero) for the adjusting factor, HYSYS automatically creates a make-up energy stream to balance the heat transfer.

Object	Description
Amount of Reaction column	Displays the rate of component that was consumed or generated during the reaction.
Adjusted Duty	Displays the amount of duty for the associate component.

8.3.4 Composition Shift Tab

The Composition Shift tab contains options to specify the composition shift affected by the design variables. These options are split into the following pages:

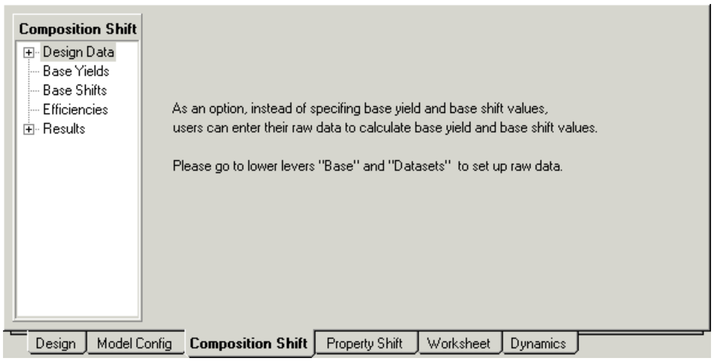
- Design Data: Base and Data
- Base Yields
- Base Shifts
- Efficiencies
- Results: Yields, Shift Extents, and Total Extents

Design Data Page

The Design Data page enables you to configure the yield of the reactor. The options are split into the following branches/pages:

- Base
- Data

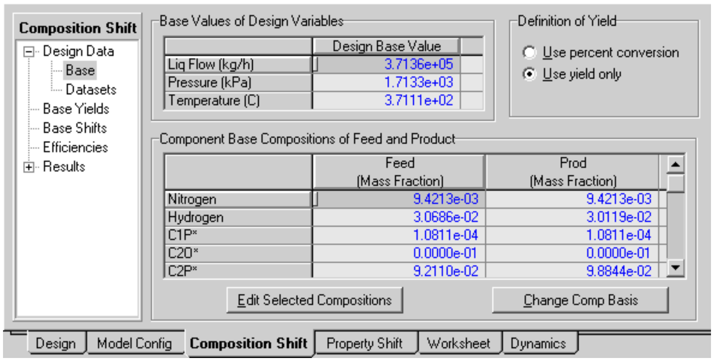
Figure 8.35



Design Data:Base Page

The Base page enables you to specify the design base values for the design variables, feed stream, and product stream. These values are use to calculate the base yield values of the components.

Figure 8.36



You can calculate base yield values using the options in the Base page or specify the base yield values in the Base Yield page.

Object	Description
Design Base Value column	Enables you to specify the design base values for the associate design variables.
Use percent conversion radio button	Enables you to model the reactor based on percent conversion specifications.
Use yield only radio button	Enables you to model the reactor based on yield specifications.
Feed column	Displays the composition of the feed stream. You can edit the composition of the stream by entering an new value in any of the cells.
Product column	Displays the composition of the product stream. You can edit the composition of the stream by entering an new value in any of the cells.
Base Conversion column	Enables you to specify the base conversion value for each component in percent value.

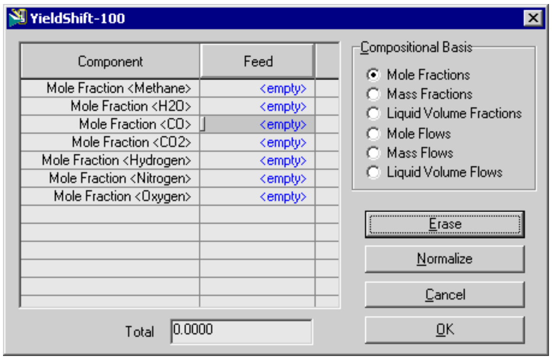
Object	Description
Edit Composition button	Enables you to edit the composition of the selected stream.
Change Comp Basis button	Enables you to change the composition basis of the selected stream.

To change the composition of a stream:

1. In the Component Base Compositions of Feed and Product group, select the stream you want to edit by clicking on a cell associated to the stream.
2. Click the **Edit Compositions** button.

   The component composition property view appears.

Figure 8.37



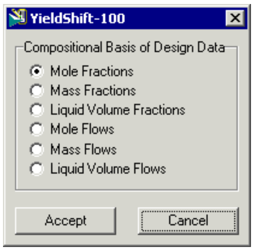
3. In the appropriate cell, enter the composition value for each component.
   - You can modify the composition basis of the stream by selecting the appropriate radio button in the Composition Basis group.
   - You can click the **Erase** button to remove all the composition values.
   - You can click the **Normalize** button to shift all the composition values so that the total value equals **1**.
   - You can click the **Cancel** button to exit the component composition property view without accepting any of the changes made.
4. Click the **OK** button to accept the modified composition values.

To change the composition basis of a stream:

1. In the Component Base Compositions of Feed and Product group, select the stream you want to edit by clicking on a cell associated to the stream.
2. Click the **Change Comp Basis** button.

   The composition basis property view appears.

Figure 8.38



3. Select the composition basis you want by clicking the appropriate radio button.

   You can click the **Cancel** button to exit the composition basis property view without accepting any of the changes made.
4. Click the **Accept** button to accept the new selection.

Calculating Base Yield Values

- Calculating the base yield values using percent conversion:

$$\text{base\_yield}_k^j = \frac{y_k^j - x_k^j \times (1 - \text{conv}_k^{base})}{\sum_{k=0}^{NC} x_k^j \times \text{conv}_k^{base}} \tag{8.21}$$

where:

base_yield_k^j = *base yield value for component k of dataset j*

y_k^j = *product stream mass fraction for component k of dataset j*

x_k^j = *feed stream mass fraction for component k of dataset j*

conv_k^{base} = *base conversion percentage value for component k*

NC = number of components

- Calculating the base yield values using yield only:

$$\mathrm{base\_yield}_k^j = y_k^j - x_k^j \tag{8.22}$$

where:

$\mathrm{base\_yield}_k^j$ = *base yield value for component k of dataset j*

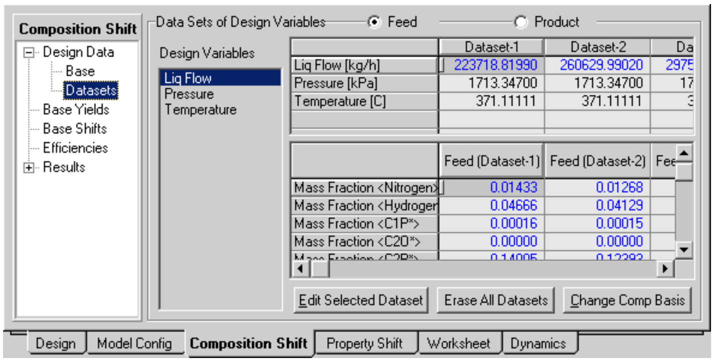
y_k^j = *product stream mass fraction for component k of dataset j*

x_k^j = *feed stream mass fraction for component k of dataset j*

Design Data:Datasets Page

The Datasets page enables you to specify the data set values for the design variables, feed stream, and product stream. These values are used to calculate the component shift for each data set of each design variable.

Figure 8.39



You can calculate base shift values using the options in the Dataset page or specify the base shift values in the Base Shifts page.

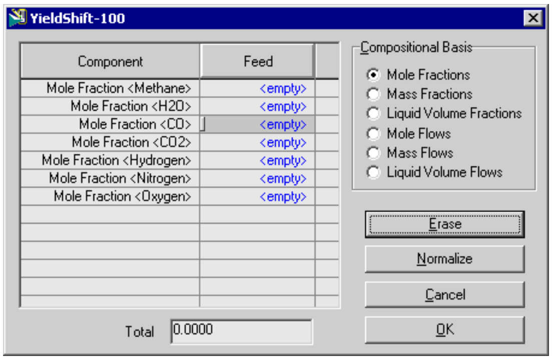
Object	Description
Feed radio button	Enables you to access and modify the composition of the feed stream.
Product radio button	Enables you to access and modify the composition of the product stream.
Design Variables list	Enables you to access and modify the data set values of the selected variable.
Dataset columns	Enables you to specify the selected design variable value for the associate data set.
Feed Dataset column	Enables you to specify the feed stream composition for the associate data set. This column is only available if the **Feed** radio button is selected.
Product Dataset column	Enables you to specify the product stream composition for the associate data set. This column is only available if the **Product** radio button is selected.
Edit Selected Dataset button	Enables you to edit the stream composition of the selected data set.
Erase All Datasets button	Deletes all the stream composition values for all data sets.
Change Comp Base button	Enables you to change the composition basis of the selected data set.

To change the stream composition of a data set:

1. In the Data Sets of Design Variables group, select the appropriate radio button to modify the feed or product stream.
2. In the Design Variables list, select the design variable associated to the stream you want to edit.
3. In the stream table, select the data set you want to modify.
4. Click the **Edit Selected Dataset** button.

The component composition property view appears.

Figure 8.40



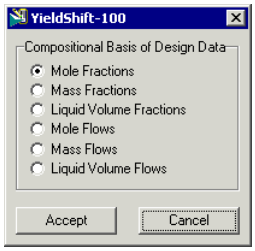
5. In the appropriate cell, enter the composition value for each component.
   - You can modify the composition basis of the stream by selecting the appropriate radio button in the Composition Basis group.
   - You can click the **Erase** button to remove all the composition values.
   - You can click the **Normalize** button to shift all the composition values so that the total value equals **1**.
   - You can click the **Cancel** button to exit the component composition property view without accepting any of the changes made.
6. Click the **OK** button to accept the modified composition values.

To change the composition basis of a data set:

1. In the Data Sets of Design Variables group, select the appropriate radio button to modify the feed or product stream.
2. In the Design Variables list, select the design variable associated to the stream you want to edit.
3. In the stream table, select the data set you want to modify.
4. Click the **Change Comp Basis** button.

The composition basis property view appears.

Figure 8.41



5. Select the composition basis you want by clicking the appropriate radio button.

   You can click the **Cancel** button to exit the composition basis property view without accepting any of the changes made.

6. Click the **Accept** button to accept the new selection.

Calculating Component Shift

The following equation is used to calculate the component shift value for the dataset of each variable.

$$\text{comp\_shift}_k^j = \frac{\text{yield}_k^{j+1} - \text{yield}_k^j}{\text{input\_var}^{j+1} - \text{input\_var}^j} \tag{8.23}$$

where:

comp_shift_k^j = *component shift value for component k of dataset j*

yield_k^{j+1} = *yield value for component k of dataset j+1*

yield_k^j = *yield value for component k of dataset j*

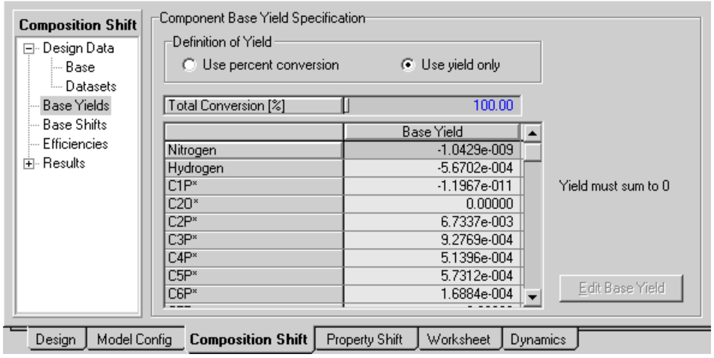
input_var^{j+1} = *maximum design variable value for dataset j+1*

input_var^j = *minimum design variable value for dataset j*

Base Yields Page

The Base Yields page enables you to specify the base yield values of each component in the reactor.

Figure 8.42



If you had already specify the yield values in the Design Data page, the base yield values in the Base Yield page will appear black and is write protected.

You can calculate base yield values using the options in the Base page or specify the base yield values in the Base Yield page.

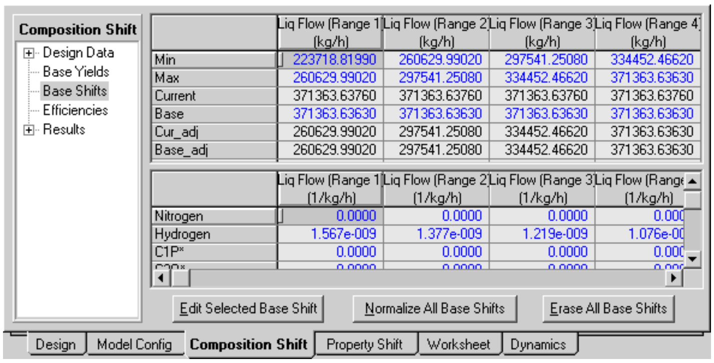
Object	Description
Use percent conversion radio button	Enables you to select the percent conversion yield type for the reactor calculation.
Use yield only radio button	Enables you to select the specify yield value type for the reactor calculation.
Base Yield column	Enables you to specify the base yield values for the components in the reactor.
Conversion [%] column	Enables you to specify the conversion percent value for the reactor. This column is only available is the **Use percent conversion** radio button is selected.

Object	Description
Total Conversion [%]	Enables you to specify in percentage the amount of components in the feed stream was converted in the reactor. This column is only available is the **Use yield only** radio button is selected.
Edit Base Yield button	Enables you to edit the component base yield values for the reactor. This button is only active if you have not specified any yield values in the **Design Data** tab.

Base Shifts Page

The Base Shifts page enables you to specify the shift values for the design parameters.

Figure 8.43



The variables in this page cannot be modified if you have already specified the values in the Design Data page.

You can calculate base shift values using the options in the Dataset page or specify the base shift values in the Base Shifts page.

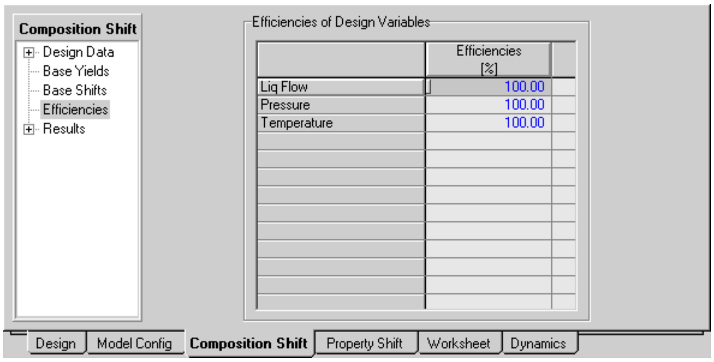
Object	Description
Min row	Enables you to specify the minimum value for the associate design parameter data set.
Max row	Enables you to specify the maximum value for the associate design parameter data set.
Current row	Displays the current value for the associate design parameter data set.

Object	Description
Base row	Enables you to specify the base value for the associate design parameter data set.
Cur_adj row	Displays the adjusted current variable value, see **Equation (8.16)**.
Base_adj row	Displays the adjusted base variable value, see **Equation (8.17)**.
Component rows	Enables you to specify the component composition value for the associate design parameter data set.
Edit Selected Base Shift button	Enables you to edit the composition of the selected design parameter data set.
Normalize All Base Shifts button	Enables you to normalize the base shifts sum to **0**.
Erase All Base Shifts button	Enables you to delete composition value for all the design parameter data sets.

Efficiencies Page

The Efficiencies page enables you to specify the percentage efficiency values of the selected design parameters.

Figure 8.44



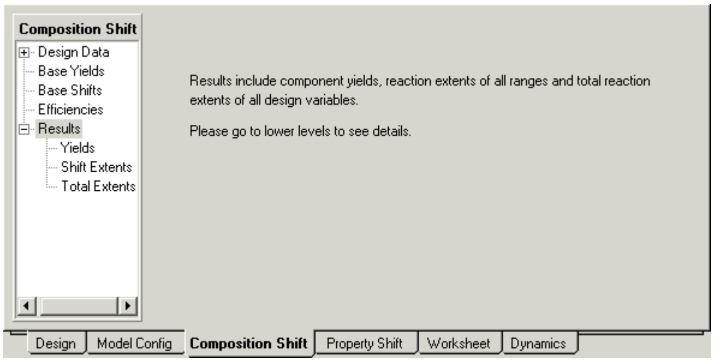
The efficiency value specified in this page is used in **Equation (8.15)** to calculate the total shift values.

Results Page

The Results page contains the calculated values from the specified parameters. The information is split into the following pages:

- Yields
- Shift Extents
- Total Extents

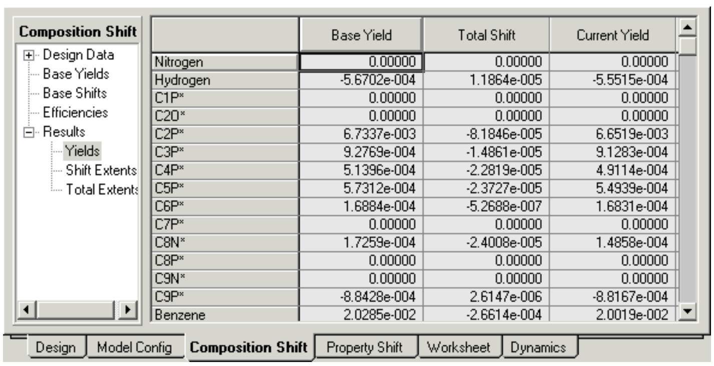
Figure 8.45



Results:Yields Page

The Yields page displays the base yield, total shift, and current yield of all the components in the reactor.

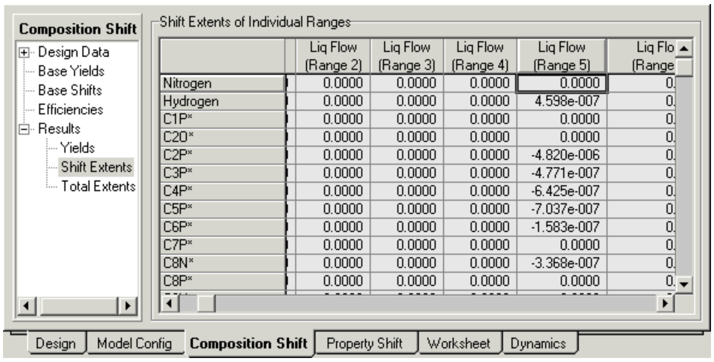
Figure 8.46



Results:Shift Extents Page

The Shift Extents page displays the component composition shift values for each data set.

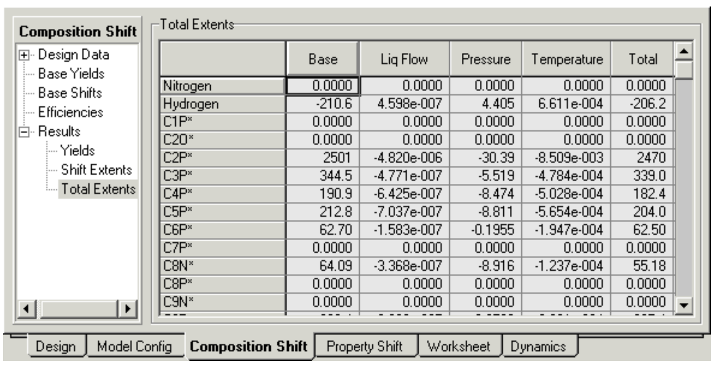
Figure 8.47



Results:Total Extents Page

The Total Extents page displays the component extent value for the base, design parameters, and total extent values.

Figure 8.48



8.3.5 Property Shift Tab

The Property Shift tab contains options to specify the property shift of the design variables. These options are split into the following pages:

- Properties
- Design Data: Base and Data
- Base Shifts
- Efficiencies
- Results: Shift Extents and Total Extents

Calculating Property Shift

The following equation is used to calculate the property shift:

$$\text{cur\_prop} = \text{base\_prop} + \text{total\_shift} \tag{8.24}$$

where:

cur_prop = *current property shift value*

base_prop = *base property value*

$$\text{total\_shift} = \sum_{i=0}^{NV} \sum_{j=0}^{NR^i} [(\text{cur\_adj}_i^j - \text{base\_adj}_i^j) \times \text{eff}_i] \tag{8.25}$$

$$\text{cur\_adj}_i^j = Max[p_i^{j,min}, Min(p_i^{j,max}, \text{cur\_value}_i)] \tag{8.26}$$

$$\text{base\_adj}_i^j = Max[p_i^{j,min}, Min(p_i^{j,max}, \text{base\_value}_i)] \tag{8.27}$$

NV = *number of input variables*

NR^i = *number of ranges for each input variable i*

eff_i = *efficiency value for design variable i, the values are user-specified in the* ***Efficiencies Page***

$p_i^{j,min}$ = *minimum range value of each dataset j from design variable i*

$p_i^{j,max}$ = *maximum range value of each dataset j from design variable i*

If the range values are not specified, HYSYS assumes negative and positive infinity values for minimum and maximum range respectively.

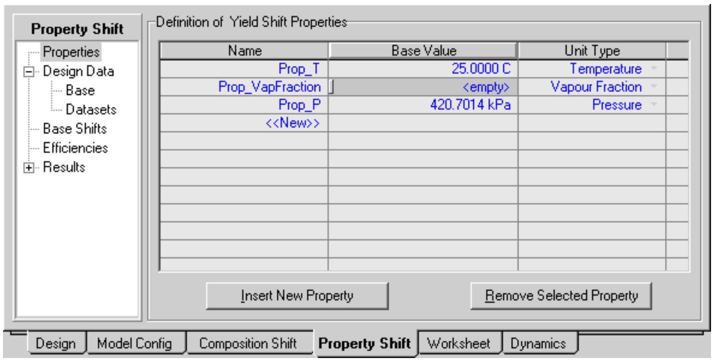
cur_value_i = *current value for design variable i*

base_value_i = *base value for design variable i*

Properties Page

The Properties page enables you to insert or remove yield shift properties.

Figure 8.49



Object	Description
Name column	Enables you to change the name of the selected property.
Value column	Enables you to specify the property value.
Unit Type column	Enables you to select the property unit type.

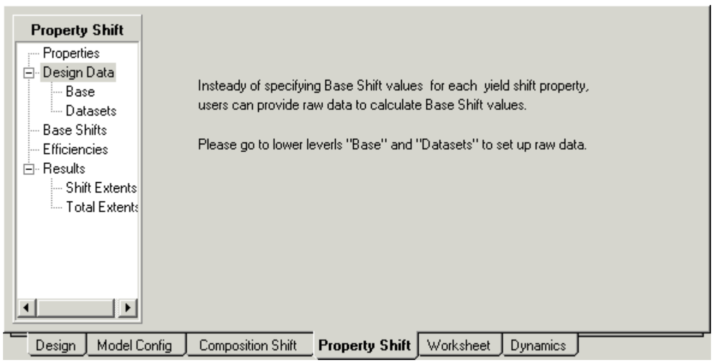
Object	Description
Insert New Property button	Enables you to add a new property.
Remove Selected Property button	Enables you to remove the selected property in the table. You can select multiple properties by pressing and holding the **CTRL** or **SHIFT** key while selecting the properties.

Design Data Page

The Design Data page enables you to configure the selected design parameter and data set. The options are split into the following branches/pages:

- Base
- Datasets

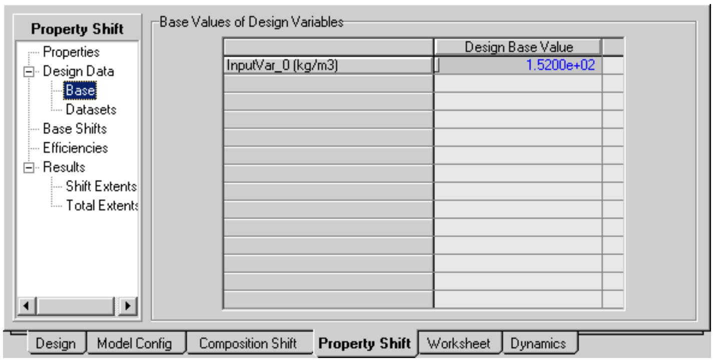
Figure 8.50



Design Data: Base Page

The Base page enables you to specify the base value for the design variables.

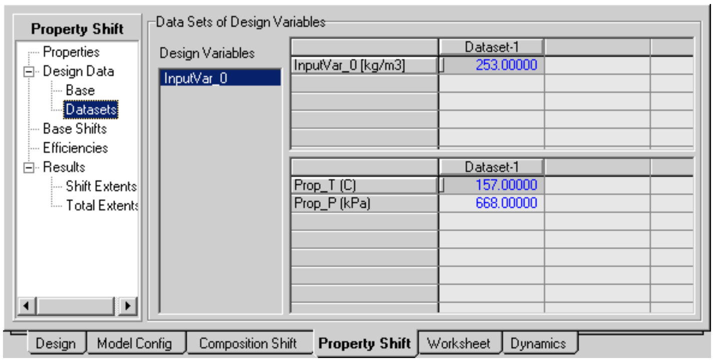
Figure 8.51



Design Data:Datasets Page

The Datasets page enables you to specify the raw data set values for the design variables and properties. These property values are used to calculate the base shift value.

Figure 8.52



Object	Description
Design Variables list	Enables you to access and modify the data set values of the selected variable.
Dataset columns	Enables you to specify the selected design variable value and the selected property values for the associate data set.

You can calculate base shift values using the options in the Dataset page or specify the base shift values in the Base Shifts page.

Calculating Property Shift Values

The following equation is used to calculate the base shift for dataset j of each variable for property shift.

$$\text{base\_shift}_k^j = \frac{\text{prop}_k^{j+1} - \text{prop}_k^j}{\text{design\_var}^{j+1} - \text{design\_var}^j} \tag{8.28}$$

The above equation is used when only raw data is supplied.

where:

base_shift_k^j = *base shift value for component k of dataset j*

prop_k^{j+1} = *property shift value for component k of dataset j+1*

prop_k^j = *property shift value for component k of dataset j*

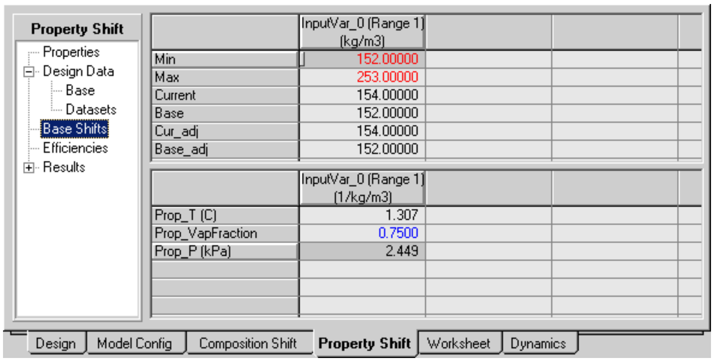
design_var^{j+1} = *maximum design variable value for dataset j+1*

design_var^j = *minimum design variable value for dataset j*

Base Shifts Page

The Base Shifts page enables you to configure the shift range and shift property value for the reactor.

Figure 8.53



You cannot modify the values in the Base Shifts page if you have already specified the values in the Properties and Design Data pages.

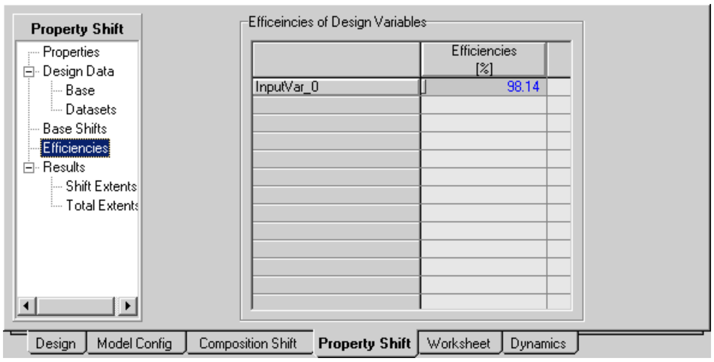
You can calculate base shift values using the options in the Dataset page or specify the base shift values in the Base Shifts page.

Object	Description
Min row	Enables you to specify the minimum value for the associate design parameter data set.
Max row	Enables you to specify the maximum value for the associate design parameter data set.
Current row	Displays the current value for the associate design parameter data set.
Base row	Enables you to specify the base value for the associate design parameter data set.
Cur_adj row	Displays the adjusted current variable value, see **Equation (8.26)**.
Base_adj row	Displays the adjusted base variable value, see **Equation (8.27)**.
Properties rows	Enables you to specify the shift property value for the associate design parameter data set.

Efficiencies Page

The Efficiencies page enables you to specify the percentage efficiency values of the selected design parameters.

Figure 8.54

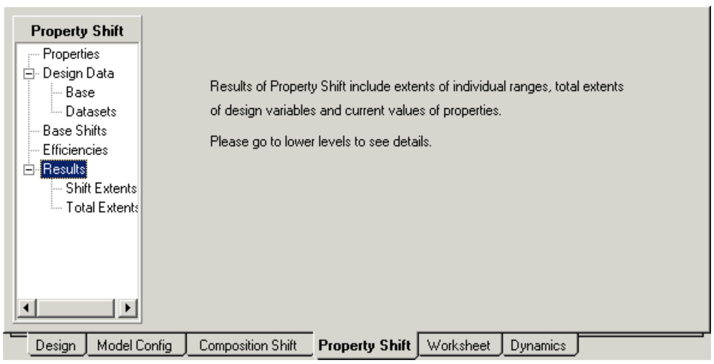


The efficiency value specified in this page is used in **Equation (8.25)** to calculate the total shift values.

Results Page

The Results page contains the calculated values for the reactor.

Figure 8.55



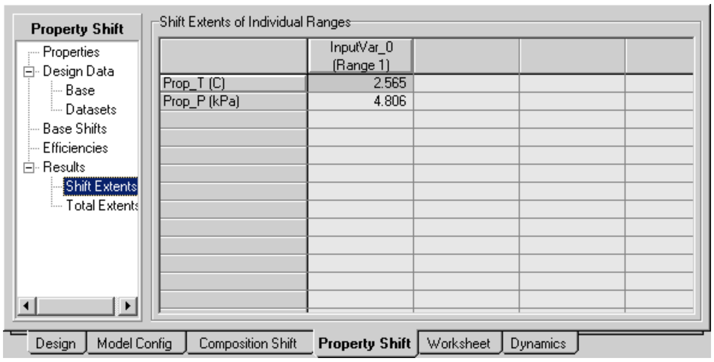
The information is split into the following branches/pages:

- Shift Extents
- Total Extents

Results:Shift Extents Page

The Shift Extents page displays the property shift values associated to the design parameters.

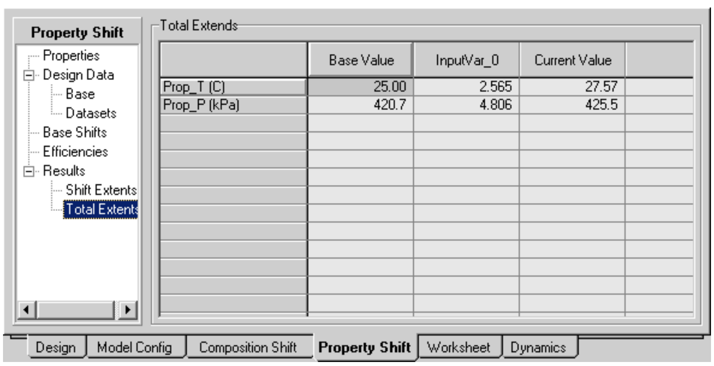
Figure 8.56



Results:Total Extents Page

The Total Extents page displays the property values for the base, design parameter, and current.

Figure 8.57



8.3.6 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

The PF Specs page is relevant to dynamics cases only.

8.3.7 Dynamics Tab

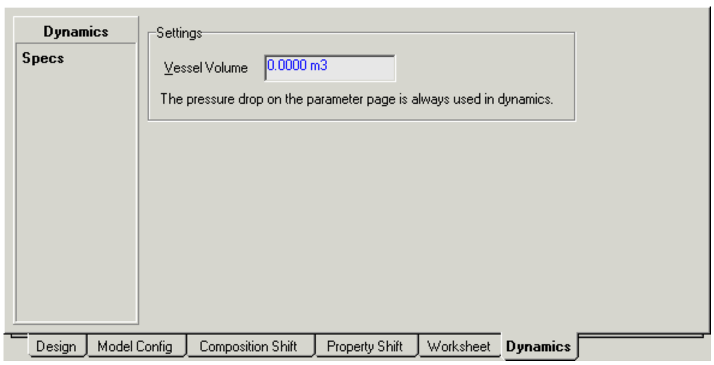
If you are working exclusively in Steady State mode, you are not required to change any information on the pages accessible through this tab.

The Dynamics tab for Yield Shift Reactor operation has only one page: Specs.

Specs Page

The Specs page contains the option to set the reactor vessel volume.

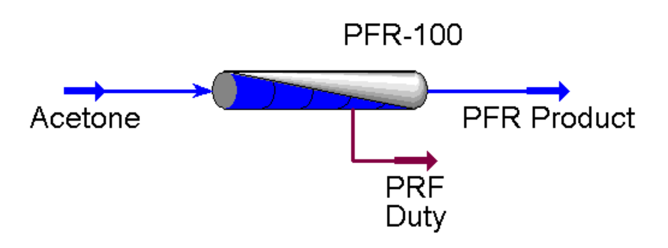
Figure 8.58



8.4 Plug Flow Reactor (PFR)

The PFR (Plug Flow Reactor, or Tubular Reactor) generally consists of a bank of cylindrical pipes or tubes. The flow field is modeled as plug flow, implying that the stream is radially isotropic (without mass or energy gradients). This also implies that axial mixing is negligible.

Figure 8.59



As the reactants flow the length of the reactor, they are continually consumed, hence, there is an axial variation in concentration. Since reaction rate is a function of concentration, the reaction rate also varies axially (except for zero-order reactions).

To obtain the solution for the PFR (axial profiles of compositions, temperature, and so forth), the reactor is divided into several subvolumes. Within each subvolume, the reaction rate is considered to be spatially uniform. A mole balance is done in each subvolume *j*:

$$F_{j0} - F_j + \int_V r_j dV = \frac{dN_j}{dt} \tag{8.29}$$

Because the reaction rate is considered spatially uniform in each subvolume, the third term reduces to $r_j V$. At steady state, the right side of this balance equals zero, and the equation reduces to:

$$F_j = F_{j0} + r_j V \tag{8.30}$$

8.4.1 Adding a Plug Flow Reactor (PFR)

There are two ways that you can add a PFR to your simulation:

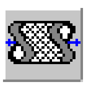
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Reactors** radio button.
3. From the list of available unit operations, select **Plug Flow Reactor**.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

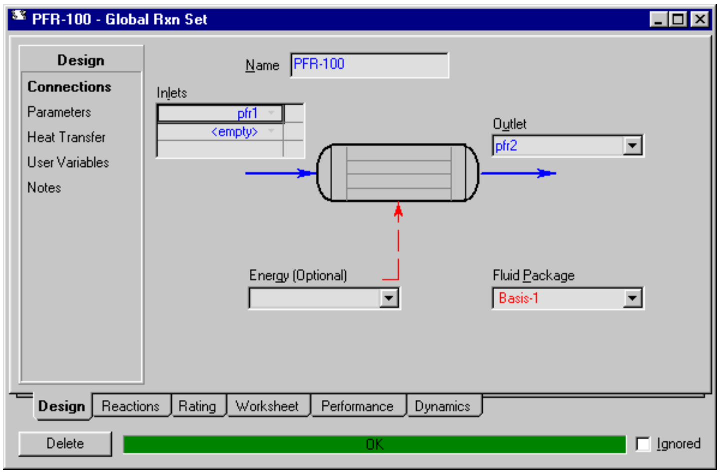
   You can also open the Object Palette by pressing **F4**.
2. Double-click the **Plug Flow Reactor** icon.



Plug Flow Reactor icon

The PFR property view appears.

Figure 8.60



8.5 Plug Flow Reactor (PFR) Property View

The Plug Flow Reactor (PFR) property view contains the following tabs:

- Design
- Reactions
- Rating
- Worksheet
- Performance
- Dynamics

8.5.1 PFR Design Tab

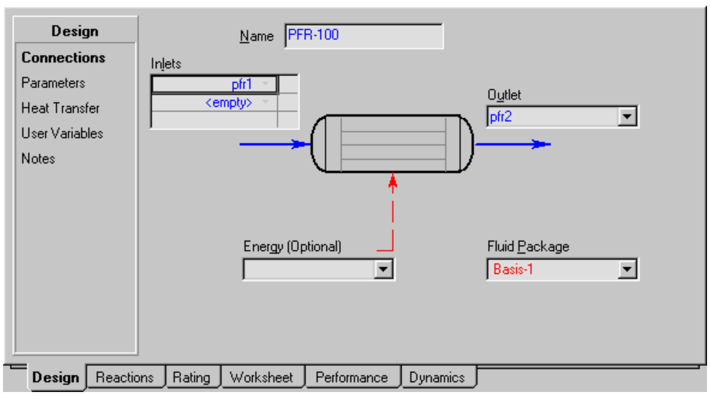
The Design tab of the PFR contains several pages, which are briefly described in the table below.

Page	Input Required
Connections	Attaches the feed and product streams to the reactor. Refer to the section below for more information.
Parameters	Allows you to specify the parameters for the pressure drops and energy streams.
Heat Transfer	Allows you to specify the heat transfer parameters.
User Variables	Allows you to create and implement User Variables.
Notes	Allows you to add relevant comments which are exclusively associated with the unit operation.

Connections Page

You can specify the name of the reactor, the feed(s) stream, product stream, and energy stream on the Connections page.

Figure 8.61



If you do not provide an energy stream, the operation is considered to be adiabatic.

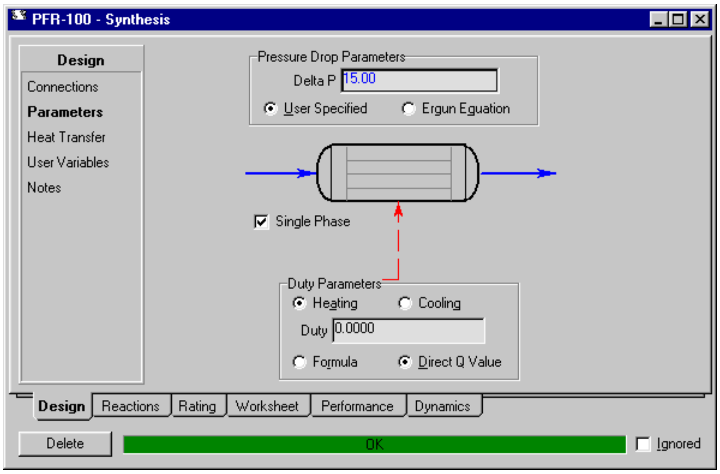
The table below describes the objects on the Connections page.

Object	Input Required
Inlets	The reactor feed stream.
Outlet	The reactor product stream.
Energy (Optional)	You are not required to provide an energy stream, however under those circumstances HYSYS assumes that the operation is adiabatic.

Parameters Page

You can instruct HYSYS on the calculations for the pressure drop and heat transfer, and also decide whether the operation is included in the calculation on the Parameters page.

Figure 8.62



Pressure Drop Parameters Group

In the Pressure Drop Parameters group, you can select one of the available radio buttons for the determination of the total pressure drop across the reactor.

The radio buttons are described in the table below.

Radio Button	Description
User Specified	Select this radio button to specify a pressure drop in the **Delta P** field.
Ergun Equation	HYSYS uses the **Ergun** equation to calculate the pressure drop across the PFR. The equation parameters include values which you specify for the PFR dimensions and feed streams: $$\frac{\Delta P g_c}{L}\frac{\varphi_s D_p}{\rho \bar{V}^2}\frac{\varepsilon^3}{1-\varepsilon} = \frac{150(1-\varepsilon)}{\varphi_s D_p \bar{V}\rho / \mu} + 1.75 \qquad (8.31)$$ *where:* ΔP = *pressure drop across the reactor* g_c = *Newton's-law proportionality factor for the gravitational force unit* L = *reactor length* φ_s = *particle sphericity* D_p = *particle (catalyst) diameter* ρ = *fluid density* $\bar{V}$ = *superficial or empty tower fluid velocity* ε = *void fraction* μ = *fluid viscosity* If you select the Ergun Equation radio button for a PFR with no catalyst (solid), HYSYS sets ΔP = 0.

When you select the Ergun Equation radio button, the Delta P field changes colour from blue to black, indicating a value calculated by HYSYS.

Duty Parameters Group

For the PFR heat transfer calculations, you can select one of the radio buttons described in the table below.

Radio Button	Description
Formula	HYSYS calculates the energy stream duty after you specify further heat transfer information on the **Heat Transfer Page**. The two fields below the radio buttons show the Energy Stream, which is attached on the Connections page, and the Calculated Duty value.
Direct Q Value	You can directly specify a duty value for the energy stream.

You can specify whether the energy stream is Heating or Cooling by selecting the appropriate radio button. This does not affect the sign of the duty stream. Rather, if the energy stream is Heating, then the duty is added to the feed. If Cooling is chosen, the duty is subtracted.

Heat Transfer Page

The format of the Heat Transfer page depends on your selection in the SS Duty Calculation Option group. There are two radio buttons:

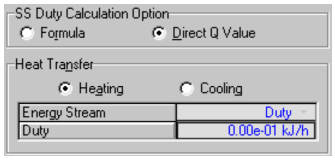
- Formula
- Direct Q Value

Your selection in the SS Duty Calculation Option group is also transferred to the Heat Transfer group on the Parameters page.

Direct Q Value Option

When you select the Direct Q Value radio button, the Heat Transfer group appears. It consists of three objects, which are described in the table below.

Figure 8.63

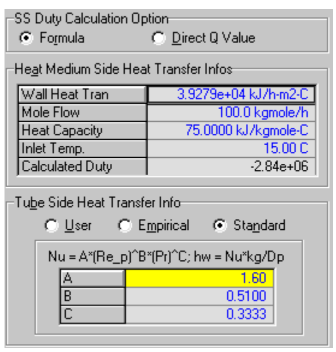


Object	Description
Energy Stream	The name of the duty stream.
Duty	The duty value to be specified in the energy stream.
Heating \ Cooling	Selecting one of these radio buttons does not affect the sign of the duty stream. Rather, if the energy stream is Heating, then the duty is added to the feed. If Cooling is chosen, the duty is subtracted.

Formula Option

When you select the Formula radio button, you instruct HYSYS to rigorously calculate the duty of each PFR subvolume using local heat transfer coefficients for the inside and the outside of each PFR tube using **Equation (8.32)** and **Equation (8.33)**.

Figure 8.64



For the Formula option, you must have an energy stream attached to the PFR. You cannot use this option while operating adiabatically.

$$Q_j = U_j A (T_{bulkj} - T_{outj}) \tag{8.32}$$

where:

Q_j = *heat transfer for subvolume j*

Resistance of the tube wall to heat transfer is neglected.

U_j = *overall heat transfer coefficient for subvolume j*

A = *surface area of the PFR tube*

T_{bulkj} = *bulk temperature of the fluid*

T_{outj} = *temperature outside of the PFR tube (utility fluid)*

$$\frac{1}{U} = \frac{1}{h_{out}} + \frac{1}{h_w} + \frac{x_w}{k_m} \tag{8.33}$$

where:

U = *overall heat transfer coefficient*

h_{out} = *local heat transfer coefficient for the outside (utility fluid)*

h_w = *local heat transfer coefficient inside the PFR tube*

$\frac{x_w}{k_m}$ = *heat transfer term for the tube wall (ignored in calculations)*

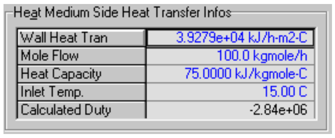
The final term in Equation (8.33), which represents the thickness of the tube divided by the thermal conductivity of the tube material, is deemed negligible and is ignored in the PFR calculations.

In each subvolume, heat is being transferred radially between the PFR fluid and the utility fluid. The two groups available on the **Heat Transfer** page allow you to specify parameters which are used in the determination of the duty.

Heat Medium Side Heat Transfer Infos Group

In the Heat Medium Side Heat Transfer Infos group, you can modify the parameters which are used to calculate the duty (Q_j) for the outside of each PFR subvolume.

Figure 8.65



If you specify a heat flow on the Energy Stream property view and select the Formula radio button on the Heat Transfer page, inconsistencies appear in the solution. You cannot specify a duty and have HYSYS calculate the same duty.

The table below describes the parameters.

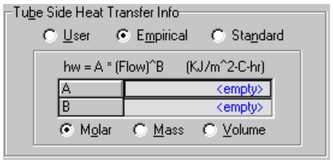
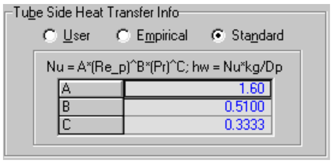
Parameter	Formula Variable	Input Required
Wall Heat Transfer Coefficient	h_{out}	Specify a value for the local heat transfer coefficient. Since the UA value, in this case the U being the local heat transfer coefficient, is constant, changes made to the specified length, diameter or number of tubes (on the Dimensions page) affects h_{out}.
Mole Flow	m	Molar flow of the energy stream utility fluid.
Heat Capacity	C_p	Heat capacity of the energy stream utility fluid.
Inlet Temperature	T	The temperature of the utility fluid entering the PFR.
Calculated Duty	Q_j	Duty calculated for each PFR subvolume.

The equation used to determine the temperature of the utility fluid entering each subvolume *j* is:

$$Q_j = m\rho C_p(T_j - T_{j+1}) \quad (8.34)$$

Tube Side Heat Transfer Info Group

In the Tube Side Heat Transfer Info group, you can select the method for determining the inside local heat transfer coefficient (h_w) by selecting one of the radio buttons and specifying the required parameters. The radio buttons are described in the table below.

Radio Button	Description	View
User	Specify a value for the local heat transfer coefficient in the User Specified input field.	
Empirical	Specify coefficients for the empirical equation which relates the heat transfer coefficient to the flowrate of the PFR fluid via the following equation: $h_w = A \times Flow^B$ (8.35) You can also choose the basis for the equation as Molar, Mass or Volume.	
Standard	Specify coefficients for the calculation of the Nusselt number, which is then used to calculate the local heat transfer coefficient: $N_u = A \times Re^B \times Pr^C$ (8.36) $h_w = \frac{N_u k_g}{D_p}$ (8.37)	

HYSYS uses the following defaults:

- A = 1.6
- B = 0.51
- C = 0.33

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page/Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

8.5.2 Reactions Tab

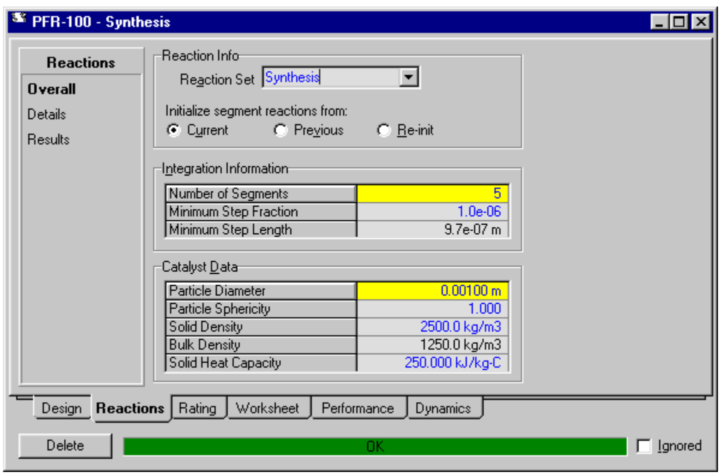
You can add a reaction set to the PFR on the Reactions tab. Notice that only Kinetic, Heterogeneous Catalytic, and Simple Rate reactions are allowed in the PFR. The tab contains the following pages:

- Overall
- Details
- Results

Overall Page

You can specify the reaction set and calculation information on the Overall page.

Figure 8.66



The Overall page consists of three groups:

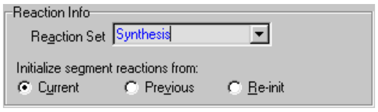
- Reaction Info
- Integration Information
- Catalyst Data

Reaction Info Group

In the Reaction Info group, you specify the following information:

- The reaction set to be used.
- The segment initialization method.

Figure 8.67



From the Reaction Set drop-down list, select the reaction set you want to use for the PFR.

The reaction set you want to use must be attached to the fluid package you are using in this environment.

As described earlier in this section, the PFR is split into segments by the reactor solver algorithm; HYSYS obtains a solution in each segment of the reactor. The segment reactions may be initialized using the following methods:

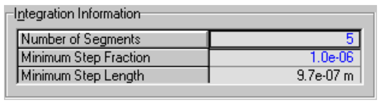
Initialization Option	Description
Current	Initializes from the most recent solution of the current segment.
Previous	Initializes from the most recent solution of the previous segment.
Re-init	Re-initializes the current segment reaction calculations.

Integration Information Group

The Integration Information group consists of three fields:

- Number of Segments
- Minimum Step Fraction
- Minimum Step Length

Figure 8.68



The table below briefly describes the fields.

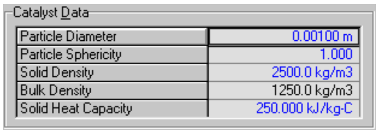
Field	Description
Number of Segments	The number of segments you want to split the PFR into.
Minimum Step Fraction	The minimum fraction an unresolved segment splits too. The length of each segment stays constant during the calculations. However, if a solution cannot be obtained for an individual segment, it is divided into smaller sections until a solution is reached. This does not affect the other segments.
Minimum Step Length	The product of the Reactor Length and the Minimum Step Fraction.

During each segment calculation, HYSYS attempts to calculate a solution over the complete segment length. If a solution cannot be obtained, the current segment is halved, and HYSYS attempts to determine a solution over the first half of the segment. The segment continues to be halved until a solution is obtained, at which point the remaining portion of the segment is calculated. If the segment is divided to the point where its length is less than the minimum step length, calculations stop.

Catalyst Data Group

If you specified a void fraction less than one on the Rating tab, the Catalyst Data group appears.

Figure 8.69



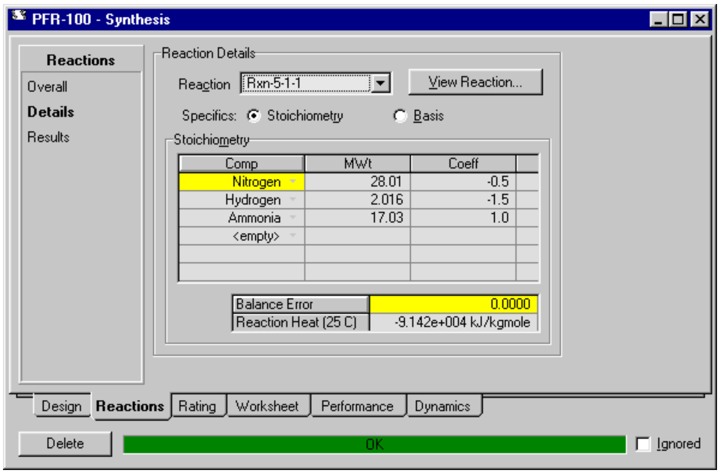
The following information must be specified:

Field	Description
Particle Diameter	The mean diameter of the catalyst particles. The default particle diameter is 0.001 m.
Particle Sphericity	This is defined as the surface area of a sphere having the same volume as the particle divided by the surface area of the particle. A perfectly spherical particle has a sphericity of 1.The Particle Diameter and Sphericity are used to calculate the pressure drop (in the Ergun pressure drop equation) if it is not specified.
Solid Density	The density of the solid portion of the particle, including the catalyst pore space (microparticle voidage).This is the mass of the particle divided by the overall volume of the particle, and therefore includes the pore space.The default is 2500 kg/m3.
Bulk Density	Equal to the solid density multiplied by one minus the void fraction. $$\rho_b = \rho_s(1-\varepsilon_{ma}) \quad (8.38)$$ *where:* ρ_b = *bulk density* ρ_s = *solid density* ε_{ma} = *macroparticle voidage (void fraction)*
Solid Heat Capacity	Used to determine the solid enthalpy holdup in dynamics.The bulk density is also required in this calculation.

Details Page

You can manipulate the reactions attached to the selected reactions set on the Details page.

Figure 8.70



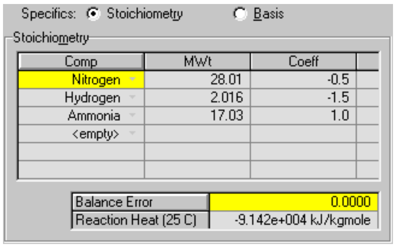
The Details page consists of three objects, which are briefly described in the table below.

Object	Description
Reaction	Allows you to select the reaction you want to use in the reactor.
View Reaction	Opens the Reaction property view for the selected reaction. This allows you to edit the reaction. Editing the Reaction propety view affects all other implementations of the selected reaction.
Specifics	Toggles between the Stoichiometry group or the Basis group (the groups are described in the following sections).

Stoichiometry Group

When you select the Stoichiometry radio button, the Stoichiometry group appears. The Stoichiometry group allows you to examine the components involved in the currently selected reaction, their molecular weights as well as their stoichiometric coefficients.

Figure 8.71



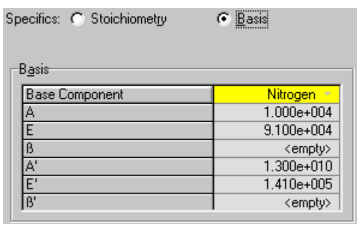
To affect change in the reaction over the entire simulation you must click the View Reaction button and make the changes in the Reaction property view.

The Balance Error (for the reaction stoichiometry) and the Reaction Heat (Heat of Reaction at 25°C) are also shown for the current reaction.

Basis Group

When you select the Basis radio button, the Basis group appears. In the Basis group, you can view the base component, and the rate expression parameters.

Figure 8.72



You can make changes to these parameters, however these changes only affect the current implementation of the reaction and are not affected by other reactors using the reaction set or reaction.

View Reaction Button

Click the **View Reaction** button to open the Reaction property view of the reaction currently selected in the Reaction drop-down list.

> **Any changes made to the Conversion Reaction property view are made globally to the selected reaction and any reaction sets which contain the reaction.**

Results Page

The Results page displays the results of a converged reactor. The page consists of the Reaction Balance group which contains two radio buttons:

- Reaction Extents
- Reaction Balance

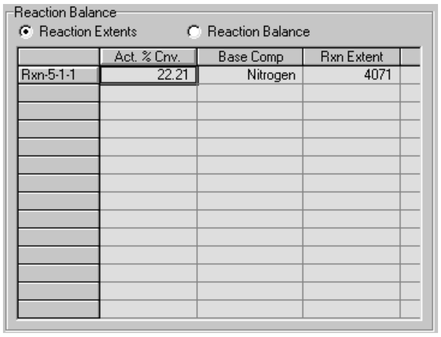
The type of results displayed varies depending on the radio button selected.

> **You can change the specified conversion for a reaction directly on this page.**

Reaction Extents

When you select the Reaction Extents radio button, the Results page appears as shown in the figure below.

Figure 8.73



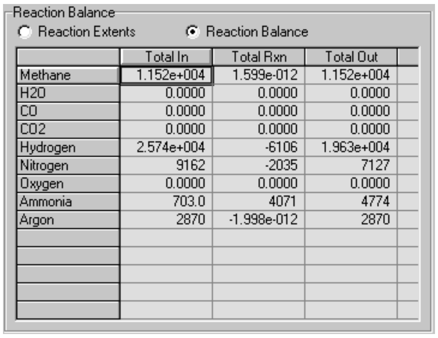
The Reaction Balance group displays the following results for a converged reactor:

Result Field	Description
Actual % Conversion	Displays the percentage of the base component in the feed stream(s) which has been consumed in the reaction.
Base Component	The reactant to which the conversion is applied.
Rxn Extent	Lists the molar rate consumption of the base component in the reaction divided by its stoichiometirc coefficient appeared in the reaction.

Reaction Balance

When you select the Reaction Balance radio button, the option provides an overall component summary for the PFR. All components which appear in the connected component list are shown.

Figure 8.74



	Total In	Total Rxn	Total Out
Methane	1.152e+004	1.599e-012	1.152e+004
H2O	0.0000	0.0000	0.0000
CO	0.0000	0.0000	0.0000
CO2	0.0000	0.0000	0.0000
Hydrogen	2.574e+004	-6106	1.963e+004
Nitrogen	9162	-2035	7127
Oxygen	0.0000	0.0000	0.0000
Ammonia	703.0	4071	4774
Argon	2870	-1.998e-012	2870

Any changes made to the global reaction affect all reaction sets to which the reaction is attached, provided local changes have not been made.

Values appear after the solution of the reactor has converged. The Total Inflow rate, the Total Reacted rate and the Total Outflow rate for each component are provided on a molar basis. Negative values indicate the consumption of a reactant, while positive values indicate the appearance of a product.

8.5.3 Rating tab

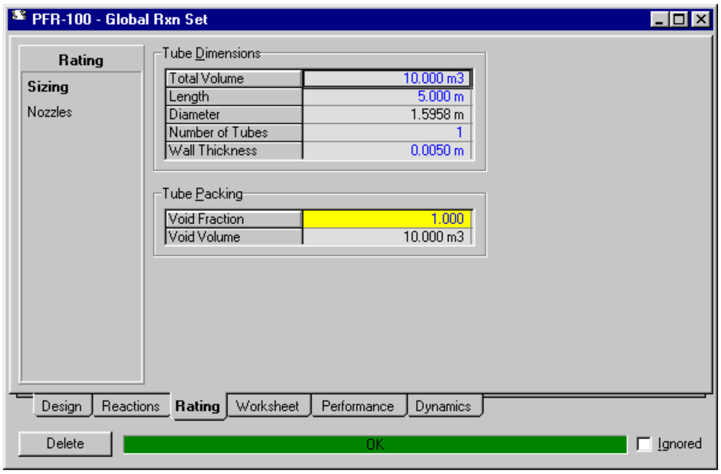
The Rating tab contains the following pages:

- Sizing
- Nozzles

Sizing Page

You can specify the tube dimensions and the tube packing information on the Sizing page.

Figure 8.75



Tube Dimensions

For the tube dimensions, you need to specify any three of the following four parameters:

Tube Dimension	Description
Total Volume	Total volume of the PFR.
Length	Total length of the individual tube.
Diameter	Diameter of an individual tube.
Number of tubes.	Total number of tubes required. This is always calculated to the nearest integer value.

When three of these dimensions are specified, the fourth is automatically calculated. Notice that the Total Volume refers to the combined volumes of all tubes.

By default, the number of tubes is set to 1. Although the number of tubes is generally specified, you can set this parameter as a calculated value by selecting the Number of Tubes field and pressing the **DELETE** key. The number of tubes are always calculated as an integer value. It is possible to obtain a rounded value of 0 as the number of tubes, depending on what you specified for the tube dimensions. In this case, you have to re-specify the tube dimensions.

The Tube Wall Thickness can also be specified.

Tube Packing

The Tube Packing group consists of two fields:

- Void Fraction
- Void Volume

The Void Volume is used to calculate the spatial velocity, which impacts the rate of reaction.

The Void Fraction is by default set to 1, in which case there is no catalyst present in the reactor. The resulting Void Volume is equal to the reactor volume.

At Void Fractions less than 1, the Void Volume is the product of the Total Volume and Void Fraction. In this case, you are also required to provide information on the Overall page of the Reactions tab. This information is used to calculate pressure drop, reactor heat capacity and spatial velocity of the fluid travelling down the reactor.

Nozzles Page

Refer to **Section 1.3.6 - Nozzles Page** for more information.

The Nozzles page contains information regarding the elevation and diameter of the nozzles.

8.5.4 Work Sheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

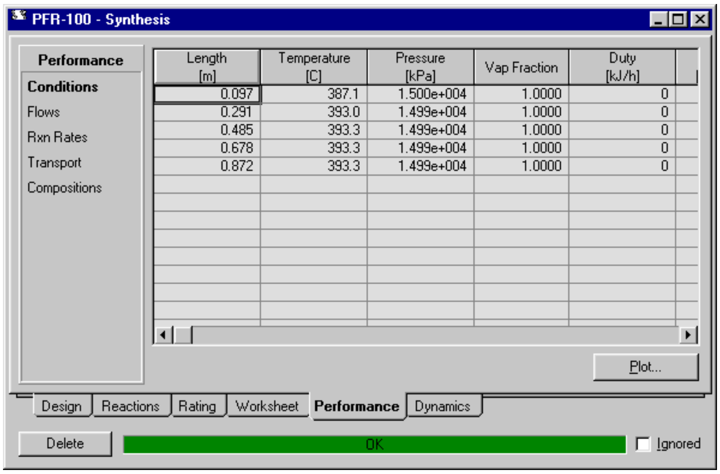
The PF Specs page is relevant to dynamics cases only.

8.5.5 Performance Tab

The Performance tab allows you to examine various axial profiles in the PFR. The tab contains the following pages, each page containing a general type of profile:

- Conditions
- Flows
- Reaction Rates (Rxn Rates)
- Transport
- Compositions

Figure 8.76



Each page consists of a table containing the relevant performance data and a Plot button which converts the data to a graphical form.

The Reactor Length is always plotted on the x-axis.

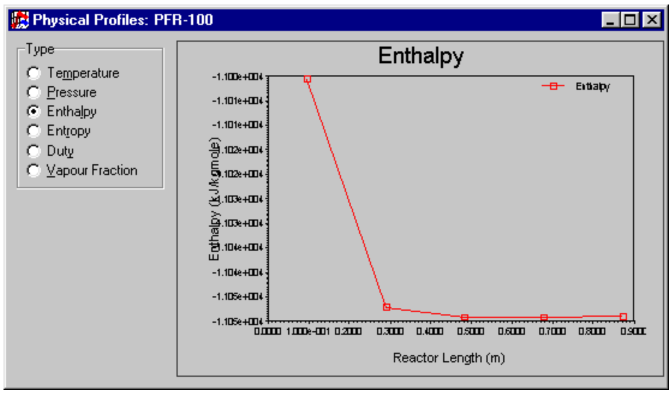
The data points are taken in the middle of each reactor segment, and correspond to the number of reactor segments you specified.

Conditions Page

The Conditions page allows you to view a table of the various physical parameters: Temperature, Pressure, Vapour Fraction, Duty, Enthalpy, Entropy, Inside HTC, and Outside HTC as a function of the Reactor Length.

If you click the Plot button, a plot similar to the one shown in the figure below appears. It shows the *selected* Physical parameter as a function of the Reactor Length.

Figure 8.77



Flows Page

There are four overall flow types which can be viewed in a table or plotted as a function of the Reactor Length:

- **Material Flow**: Molar, Mass, or Volume
- **Energy**: Heat

If you click the **Plot** button, the table appears in a graphical form.

Reaction Rates Page

You can view either Reaction Rate or Component Production Rate data as a function of the Reactor Length on the Rxn Rates page. You can toggle between the two data sets by selecting the appropriate radio button.

Although only one reaction set can be attached to the PFR, it can contain multiple reactions.

You can view the data in graphical form by clicking the Plot button.

Transport Page

The overall Transport properties appear in a tabular form as a function of the Reactor Length on the Transport page.

Transport Properties:

- Viscosity
- Molar Weight
- Mass Density
- Heat Capacity
- Surface Tension
- Z Factor

You can view the data in a graphical form by clicking the Plot button. Select the appropriate radio button to display the selected plot.

Compositions Page

You can view individual component profiles using one of six composition bases:

- Molar Flow
- Mass Flow
- Liquid Volume Flow
- Fraction:
  - Mole Fraction
  - Mass Fraction
  - Liquid Volume Fraction

You can display the data in a plot form by clicking the Plot button.

8.5.6 Dynamics Tab

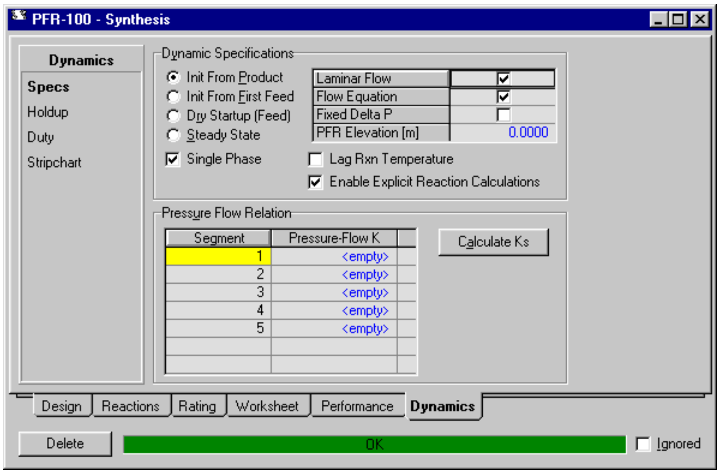
The Dynamics tab contains the following pages:

- Specs
- Holdup
- Duty
- Stripchart

If you are working exclusively in Steady State mode, you are not required to change any information on the pages accessible through the Dynamics tab.

Specs Page

Figure 8.78



Dynamic Specifications Group

The Dynamic Specifications group consists of eleven objects, which are described in the table below.

Objects	Description
Initialize from Products radio button	The composition of the holdup is calculated from a weighted average of all products exiting the holdup. A PT flash is performed to determine other holdup conditions.
Initialize from First Feed radio button	The composition of the holdup is calculated from the first feed entering the PFR reactor.
Dry Startup radio button	The calculations based on the holdup starts with no fluid in it.
Steady State radio button	Uses steady state results to initialize the holdup.
Single Phase checkbox	Allows you to specify a single phase reaction. Otherwise HYSYS considers it a vapour-liquid reaction.
Laminar Flow checkbox	Assumes laminar flow in the PFR.

Objects	Description
Flow Equation checkbox	Uses the flow equation to calculate the pressure gradient across the PFR. You are required to either estimate k values in steady state (by clicking the Calculate K’s button) or specifying your own values in the Pressure Flow Relation group.
Fixed Delta P checkbox	Assumes a constant pressure drop across the PFR. Does not require k values.
PFR Elevation cell	The height above ground that the PFR is currently positioned.
Lag Rxn Temperature checkbox	The option is designed to speed up the dynamic run for the reaction solver when the run has to invoke the steady state reaction solver. Mathematically, when you select the **Lag Rxn Temperature** checkbox, the reaction solver flashes with the explicit Euler method. Otherwise, for a dynamic run, the steady state reaction solver always flashes with the implicit Euler methods which could be slow with many iterations. The Lag Rxn Temperature may cause some instability due to the nature of the explicit Euler method. But it must compromise with the dynamic step size.
Enable Explicit Reaction Calculation checkbox	The Enable Explicit Reaction Calculations is defaulted to be used for dynamic run reaction solver. The explicit reaction solver is quick, but can introduce instability. You can deactivate this option. The implicit reaction solver is used instead.

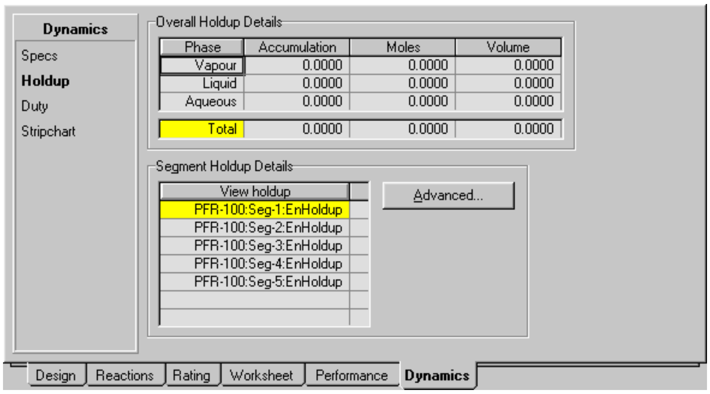
Pressure Flow Relation Group

The Pressure Flow Relation group consists mainly of a table of the k values for each segment in the PFR. You can enter your own k values into this table or, while you are in Steady State mode, you can click the Calculate K’s button and HYSYS calculates the k values using the steady state data.

Holdup Page

The Holdup page contains information regarding the properties, composition, and amount of the holdup in each phase in the PFR.

Figure 8.79



The Holdup page consists of two groups:

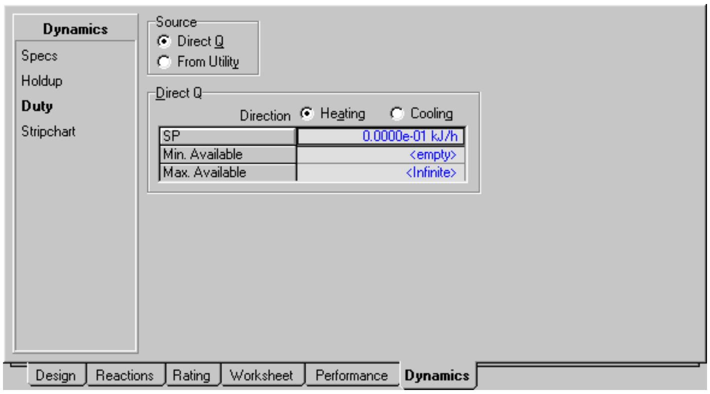
Refer to **Section 1.3.3 - Holdup Page** for more information.

- Overall Holdup Details group displays the holdup data within the PFR operation.
- Segment Holdup Details enables you to select and view individual holdup data of all the segments in the PFR operation.

Duty Page

In the Source group, you can choose whether HYSYS calculates the duty applied to the vessel from a Direct Q or a Utility.

Figure 8.80



If you select the Direct Q radio button, you can directly specify the duty applied to the holdup in the SP field.

For more information regarding how the utility option calculates duty, refer to **Chapter 5 - Logical Operations**.

If you select the Utility radio button, you can specify the flow of the utility fluid. The duty is then calculated using the local overall heat transfer coefficient, the inlet fluid conditions, and the process conditions. The calculated duty is then displayed in the SP field or the Heat Flow field.

If you select the Heating radio button, the duty shown in the SP field or Heat Flow field is added to the holdup. If you select the Cooling radio button, the duty is subtracted from the holdup.

Stripchart Page

Refer to **Section 1.3.7 - Stripchart Page/Tab** for more information.

The Stripchart page allows you to select and create default strip charts containing various variable associated to the operation.

9 Rotating Operations

9.1 Centrifugal Compressor or Expander

The Centrifugal Compressor operation is used to increase the pressure of an inlet gas stream with relative high capacities and low compression ratios. Depending on the information specified, the Centrifugal Compressor calculates either a stream property (pressure or temperature) or a compression efficiency.

A Centrifugal Compressor can also be used to represent a Pump operation when a more rigorous pump calculation is required. The Pump operation in HYSYS assumes that the liquid is incompressible. Therefore, if you want to pump a fluid near its critical point (where it becomes compressible), you can do so by representing the Pump with a Centrifugal Compressor. The Centrifugal Compressor operation takes into account the compressibility of the liquid, thus performing a more rigorous calculation.

The Expander operation is used to decrease the pressure of a high pressure inlet gas stream to produce an outlet stream with low pressure and high velocity. An expansion process involves converting the internal energy of the gas to kinetic energy and finally to shaft work. The Expander calculates either a stream property or an expansion efficiency.

There are several methods for the Centrifugal Compressor or Expander to solve, depending on what information has been specified, and whether or not you are using the compressor’s characteristic curves. In general, the solution is a function of flow, pressure change, applied energy, and efficiency. The Centrifugal Compressor or Expander provides a great deal of flexibility with respect to what you can specify and what it then calculates. You must ensure that you do not enable too many of the solution options or inconsistencies may result.

The operating characteristics curves of a compressor is usually expressed as a set of polytropic head and efficiency curves made by manufacturers.

Some of the features in the dynamic Centrifugal Compressor and Expander operations include:

- Dynamic modeling of friction loss and inertia in the Centrifugal Compressor or Expander.
- Dynamic modeling which supports shutdown and startup behaviour.
- Multiple head and efficiency curves.
- Modeling of Stonewall and Surge conditions of the Centrifugal Compressor or Expander.
- A dedicated surge controller which features quick opening capabilities.
- Handling of phase changes that may occur in the unit operation (for example Expanders producing liquid).
- Linking capabilities with other rotational equipment operating at the same speed with one total power.

Typical Solution Methods

Without Curves	With Curves
1. Flow rate and inlet pressure are known. 2. Specify outlet pressure. 3. Specify either Adiabatic or Polytropic efficiency. 4. HYSYS calculates the required energy, outlet temperature, and other efficiency.	1. Flow rate and inlet pressure are known. 2. Specify operating speed. 3. HYSYS uses curves to determine efficiency and head. 4. HYSYS calculates outlet pressure, temperature, and applied duty.
1. Flow rate and inlet pressure are known. 2. Specify efficiency and duty. 3. HYSYS calculates outlet pressure, temperature, and other efficiency.	1. Flow rate, inlet pressure, and efficiency are known. 2. HYSYS interpolates curves to determine operating speed and head. 3. HYSYS calculates outlet pressure, temperature, and applied duty.

The thermodynamic principles governing the Centrifugal Compressor and Expander operations are the same, but the direction of the energy stream flow is opposite. Compression requires energy, while expansion releases energy.

9.1.1 Theory

Steady State

For a Centrifugal Compressor, the isentropic efficiency is given as the ratio of the isentropic (ideal) power required for compression to the actual power required:

$$Efficiency(\%) = \frac{Power\ Required_{isentropic}}{Power\ Required_{actual}} \times 100\% \tag{9.1}$$

Throughout this chapter you will see “isentropic” and “adiabatic” used interchangeably. This is because they are the same.

For an Expander, the efficiency is given as the ratio of the actual power produced in the expansion process to the power produced for an isentropic expansion:

$$Efficiency(\%) = \frac{Fluid\ Power\ Produced_{actual}}{Fluid\ Power\ Produced_{isentropic}} \times 100\% \tag{9.2}$$

For an adiabatic Centrifugal Compressor and Expander, HYSYS calculates the centrifugal compression (or expansion) rigorously by following the isentropic line from the inlet to outlet pressure. Using the enthalpy at that point, as well as the specified efficiency, HYSYS then determines the actual outlet enthalpy. From this value and the outlet pressure, the outlet temperature is determined.

For a polytropic Centrifugal Compressor or Expander, the path of the fluid is neither adiabatic nor isothermal. For a 100% efficient process, there is only the condition of mechanical reversibility. For an irreversible process, the polytropic efficiency is less than 100%. Depending on whether the process is an expansion or compression, the work determined for the mechanically reversible process is multiplied or divided by an efficiency to give the actual work. The form of the polytropic efficiency equations are the same as **Equation (9.1)** and **Equation (9.2)**.

Notice that all intensive quantities are determined thermodynamically, using the specified Property Package. In general, the work for a mechanically reversible process can be determined from:.

$$W = \int VdP \tag{9.3}$$

where:

W = work

V = volume

dP = pressure difference

As with any unit operation, the calculated information depends on the information which is specified by the user. In the case where the inlet and outlet pressures and temperatures of the gas are known, the ideal (isentropic) power of the Operation is calculated using one of the above equations, depending on the Centrifugal Compressor or Expander type. The actual power is equivalent to the heat flow (enthalpy) difference between the inlet and outlet streams.

- For the Centrifugal Compressor:

$$\text{Power Required}_{\text{actual}} = \text{Heat Flow}_{\text{outlet}} - \text{Heat Flow}_{\text{inlet}} \tag{9.4}$$

  where the efficiency of the Centrifugal Compressor is then determined as the ratio of the isentropic power to the actual power required for compression.

- For the Expander:

$$\text{Power Produced}_{\text{actual}} = \text{Heat Flow}_{\text{inlet}} - \text{Heat Flow}_{\text{outlet}} \tag{9.5}$$

  The efficiency of the Expander is then determined as the ratio of the actual power produced by the gas to the isentropic power.

In the case where the inlet pressure, the outlet pressure, the inlet temperature and the efficiency are known, the isentropic power is once again calculated using the appropriate equation. The actual power required by the Centrifugal Compressor (enthalpy difference between the inlet and outlet streams) is calculated by dividing the ideal power by the compressor efficiency. The outlet temperature is then rigorously determined from the outlet enthalpy of the gas using the enthalpy expression derived from the property method being used. For an isentropic compression or expansion (100% efficiency), the outlet temperature of the gas is always lower than the outlet temperature for a real compression or expansion.

Dynamic

An essential concept associated with the Centrifugal Compressor and Expander operations is the isentropic and polytropic power. The calculation of these parameters and other quantities are taken from "Compressors and Exhausters - Power Test Codes" from the American Society of Mechanical Engineers.

The isentropic or polytropic power, W, can be calculated from:

$$W = F_1(MW)\left(\frac{n}{n-1}\right)CF\left(\frac{P_1}{\rho_1}\right) \times \left[\left(\frac{P_2}{P_1}\right)^{\left(\frac{n-1}{n}\right)} - 1\right] \tag{9.6}$$

where:

n = volume exponent

CF = correction factor

P_1 = pressure of the inlet stream

P_2 = pressure of the exit stream

ρ_1 = density of the inlet stream

F_1 = molar flow rate of the inlet stream

MW = molecular weight of the gas

Isentropic power is calculated by defining the volume exponent as:

$$n = \frac{\ln(P_2 / P_1)}{\ln(\rho'_2 / \rho_1)} \tag{9.7}$$

where:

ρ'_2 = *density of the exit stream corresponding to the inlet entropy*

Polytropic power is calculated by defining the volume exponent as:

$$n = \frac{\ln(P_2 / P_1)}{\ln(\rho_2 / \rho_1)} \tag{9.8}$$

where:

ρ_2 = *density of the exit stream*

The correction factor is calculated as:

$$CF = \frac{h'_2 - h_1}{\left(\frac{n}{n-1}\right)\left(\frac{P_2}{\rho'_2} - \frac{P_1}{\rho_1}\right)} \tag{9.9}$$

where:

h'_2 = *enthalpy of the exit stream corresponding to the inlet entropy*

h_1 = *enthalpy of the inlet stream*

An isentropic flash is performed to calculate the values of h'_2 and ρ'_2.

HYSYS calculates the compression (or expansion) rigorously by following the isentropic line from the inlet to the exit pressure. The path of a polytropic process is neither adiabatic nor isothermal. The only condition is that the polytropic process is reversible.

Equations Used

The Centrifugal Compressor equations are used for the Centrifugal Compressor. The Expander equations are used for the Expander.

Compressor Efficiencies

The Adiabatic and Polytropic Efficiencies are included in the Centrifugal Compressor calculations. An isentropic flash (P_{in} and $Entropy_{in}$) is performed internally to obtain the ideal (isentropic) properties.

Expander Efficiencies

For an Expander, the efficiencies are parts of the Expander calculations, and an isentropic flash is performed as well. The flash is done on the Expander fluid, and the results are not stored.

Efficiencies	Compressor	Expander
Adiabatic	$\frac{\text{Work Required}_{(ideal)}}{\text{Work Required}_{(actual)}} = \frac{(H_{out} - H_{in})_{(ideal)}}{(H_{out} - H_{in})_{(actual)}}$	$\frac{\text{Work Produced}_{(actual)}}{\text{Work Produced}_{(ideal)}} = \frac{(H_{out} - H_{in})_{(actual)}}{(H_{out} - H_{in})_{(ideal)}}$
Polytropic	$\frac{\left[\left(\frac{P_{out}}{P_{in}}\right)^{\left(\frac{n-1}{n}\right)} - 1\right] \times \left[\left(\frac{n}{(n-1)}\right) \times \left(\frac{k-1}{k}\right)\right]}{\left[\left(\frac{P_{out}}{P_{in}}\right)^{\left(\frac{k-1}{k}\right)} - 1\right]} \times AdiabaticEff$ *where:* $n = \frac{\log(P_{out}/P_{in})}{\log(\rho_{out,\ actual}/\rho_{in})}$ $k = \frac{\log(P_{out}/P_{in})}{\log(\rho_{out,\ ideal}/\rho_{in})}$	$\frac{\left[\left(\frac{P_{out}}{P_{in}}\right)^{\left(\frac{k-1}{k}\right)} - 1\right]}{\left[\left(\frac{P_{out}}{P_{in}}\right)^{\left(\frac{n-1}{n}\right)} - 1\right] \times \left[\left(\frac{n}{(n-1)}\right) \times \left(\frac{k-1}{k}\right)\right]} \times AdiabaticEff$ *where:* $n = \frac{\log(P_{out}/P_{in})}{\log(\rho_{out,\ actual}/\rho_{in})}$ $k = \frac{\log(P_{out}/P_{in})}{\log(\rho_{out,\ ideal}/\rho_{in})}$

Efficiencies	Compressor	Expander
	where:	*P = pressure*
	H = mass enthalpy	*ρ = mass density*
	out = product discharge	*n = polytropic exponent*
	in = feed stream	*k = isentropic exponent*

Compressor Heads

The Adiabatic and Polytropic Heads are performed after the Centrifugal Compressor calculations are completed, only when the Results page of the Centrifugal Compressor is selected. The Work Required (actual) is the compressor energy stream (heat flow). The Polytropic Head is calculated based on the ASME method ("The Polytropic Analysis of Centrifugal Compressors", Journal of Engineering for Power, J.M. Schultz, January 1962, p. 69-82).

Expander Heads

The Adiabatic and Polytropic Heads are performed after the Expander calculations are completed, only when the Results page of the Expander is selected. The Work Produced (actual) is the Expander energy stream (heat flow).

Head	Compressor	Expander
Adiabatic	$\frac{Work\ Required_{(actual)}}{MassFlowRate} \times AdiabaticEff \times \frac{1}{(g/g_c)}$	$\frac{Work\ Produced_{(actual)}}{MassFlowRate} \times \frac{1}{AdiabaticEff} \times \frac{1}{(g/g_c)}$
Polytropic	$f \times \left(\frac{n}{n-1}\right) \times \left[\left(\frac{P_{out}}{\rho_{out,actual}}\right) - \left(\frac{P_{in}}{\rho_{in}}\right)\right] \times \frac{1}{(g/g_c)}$ *where:* $f = \frac{H_{out,ideal} - H_{in}}{\left(\frac{k}{k-1}\right) \times \left[\left(\frac{P_{out}}{\rho_{out,ideal}}\right) - \left(\frac{P_{in}}{\rho_{in}}\right)\right]}$ $n = \frac{\log(P_{out}/P_{in})}{\log(\rho_{out,\ actual}/\rho_{in})}$ $k = \frac{\log(P_{out}/P_{in})}{\log(\rho_{out,\ ideal}/\rho_{in})}$	$-f \times \left(\frac{n}{n-1}\right) \times \left[\left(\frac{P_{out}}{\rho_{out,actual}}\right) - \left(\frac{P_{in}}{\rho_{in}}\right)\right] \times \frac{1}{(g/g_c)}$ *where:* $f = \frac{H_{out,ideal} - H_{in}}{\left(\frac{k}{k-1}\right) \times \left[\left(\frac{P_{out}}{\rho_{out,ideal}}\right) - \left(\frac{P_{in}}{\rho_{in}}\right)\right]}$ $n = \frac{\log(P_{out}/P_{in})}{\log(\rho_{out,\ actual}/\rho_{in})}$ $k = \frac{\log(P_{out}/P_{in})}{\log(\rho_{out,\ ideal}/\rho_{in})}$
	where: *H = mass enthalpy* *out = product discharge* *in = feed stream* *P = pressure*	*ρ = mass density* *f = polytropic head factor* *n = polytropic exponent* *k = isentropic exponent*

9.1.2 Compressor or Expander Property View

There are two ways that you can add a Compressor or Expander to your simulation:

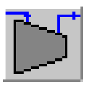
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

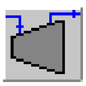
   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Rotating Equipment** radio button.
3. From the list of available unit operations, select **Compressor** or **Expander**.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

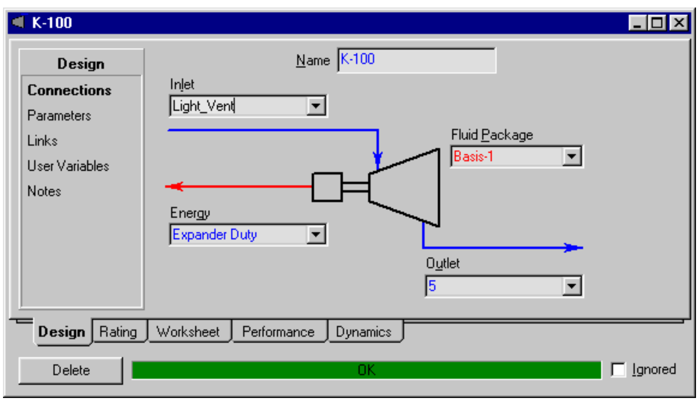
   You can also open the Object Palette by pressing **F4**.
2. Double-click the **Compressor** icon or **Expander** icon.


Compressor icon


Expander icon

The Compressor or Expander property view appears.

Figure 9.1



9.1.3 Design Tab

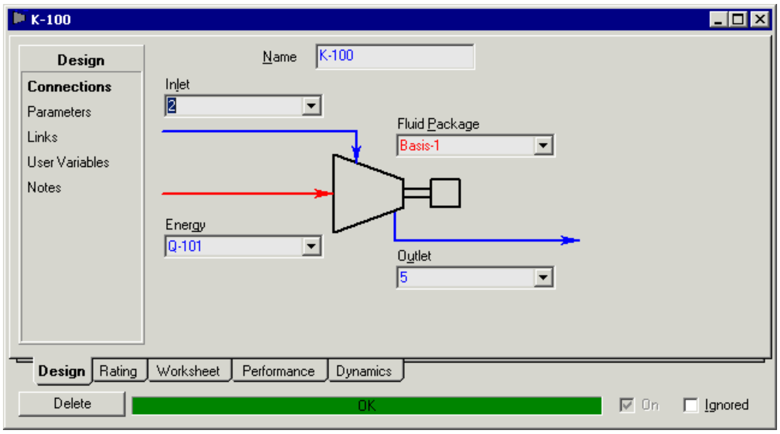
The Design tab contains the following pages:

- Connections
- Parameters
- Links
- User Variables
- Notes

Connections Page

The figure below shows the Connections page for the Centrifugal Compressor property view.

Figure 9.2



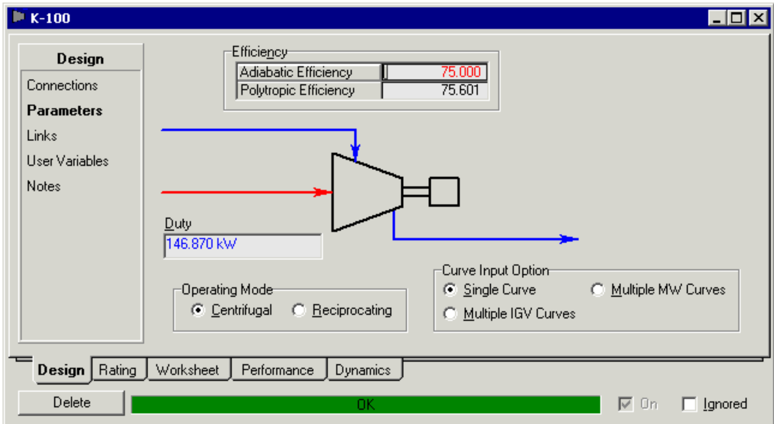
The Connections page allows you to specify the name of the operation, as well as the inlet stream, outlet stream, and energy stream.

The information required on the Connections page of the Expander is identical; the only difference is that the Expander icon is shown rather than the Compressor icon.

Parameters Page

You can specify the duty of the attached energy stream on the Parameters page, or allow HYSYS to calculate it.

Figure 9.3



The adiabatic and polytropic efficiencies appear on this page as well.

You can specify only one efficiency, either adiabatic or polytropic. If you specify one efficiency and a solution is obtained, HYSYS back calculates the other efficiency, using the calculated duty and stream conditions.

The differences between the Parameters page for the Compressor and the Expander are the Operating Mode group and Curve Input Option group that are present only in the compressor property view.

The options in the Operating Mode and Curve Input Option groups are only available for the Compressor unit operation.

In the Operating Mode group, you can switch between a Centrifugal and Reciprocating Compressor by selecting the corresponding radio button.

If you choose the Centrifugal radio button, the radio buttons in the Curve Input Option group are enabled.

In the Curve Input Option group the following radio buttons are available:

- Select the **Single Curve** radio button, to model your compressor with a single pair of head vs. flow and efficiency vs. flow curves.

The Single Curve radio button still allows multiple curves as a function of speed, but not MW or IGV position.

- Select the **Multiple MW Curves** radio button, if you have a set of curves that describe the compressor performance as a function of the flowing gas molecular weight (MW).
- Select the **Multiple IGV Curves** radio button, if you have a set of curves that describe the compressor performance as a function of inlet guide vane (IGV) position.

When you select Multiple IGV Curves radio button, the current inlet guide vane position is specified during the operation of the compressor.

Links Page

Compressors and expanders modeled in HYSYS can have shafts that are physically connected to the unit operation. Linking compressors and expanders in HYSYS means the:

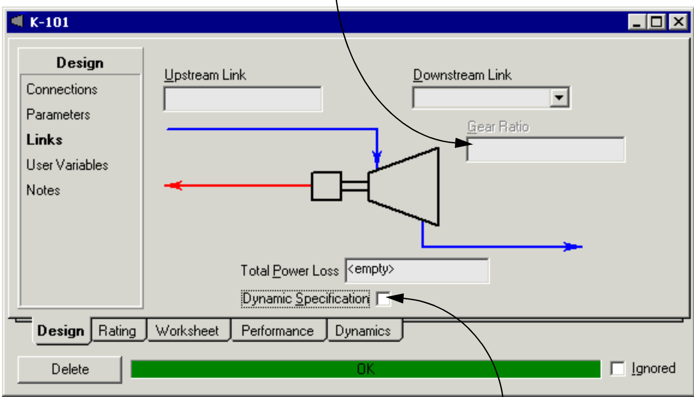
- Speed of each linked unit operation is the same.
- Sum of the duties of each linked Compressor or Expander and the total power loss equals zero.

The rotational linker operates both in Steady State and Dynamic mode.

It is not significant which order the Compressors or Expanders are linked. The notion of upstream and downstream links is arbitrary and determined by the user.

Figure 9.4

The Gear Ratio field displays the ratio of the speed from the next linked operation divided by the speed of the current pump.



Select the **Dynamic Specification** checkbox to specify the total power loss (or power gain by entering a negative value) for the linked operation.

You can ignore this option in Steady State mode.

A list of available compressors or expanders can be displayed by clicking the down arrow  in the **Downstream Link** field. In most cases, one additional specification for any of the linked operations is required to allow the simulation case to completely solve.

Ideally, you should specify one of the following for any of the linked unit operations.

- Duty
- Speed
- Total Power Loss

It is also possible to link an Expander to a Compressor, and use the Expander to generate kinetic energy to drive the Compressor. If this option is chosen, the total power loss is typically specified as zero.

Dynamics Mode

In Dynamics mode, at least one curve must be specified in the Curves page of the Rating tab for each linked unit operation. Ideally, a set of linked compressor or expanders should only have the **Use Characteristic Curves** checkbox selected in the Specs page of the Dynamics tab. In addition, the total power loss for the linked operations should be specified. Usually, total power input to the linked compressors or expanders is calculated in a Spreadsheet operation and specified by you in the Total Power Loss field.

If you want to provide the total power input to a set of linked compressors or expanders, the total power input to the linked operations is defined in terms of a total power loss. The relationship is as follows:

$$\text{Total Power Input} = -\text{Total Power Loss} \qquad (9.10)$$

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

9.1.4 Rating Tab

The Rating tab contains following pages:

- Curves
- Flow Limits
- Nozzles

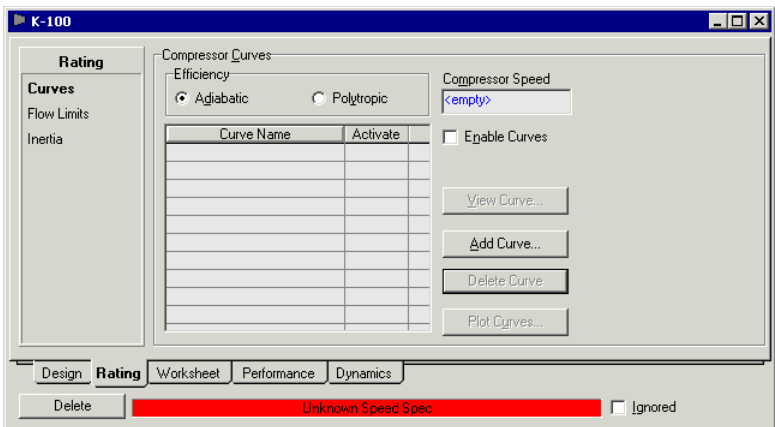
The Nozzles page is only visible if you have the HYSYS Dynamics license.

- Inertia

Curves Page

One or more Centrifugal Compressor or Expander curves can be specified on the Curves page.

Figure 9.5



You can create adiabatic or polytropic plots for values of efficiency and head. The efficiency and head for a specified speed can be plotted against the capacity of the Centrifugal Compressor or Expander. Multiple curves can be plotted to show the dependence of efficiency and head on the speed of Centrifugal Compressors or Expanders.

If you do not use curves, specify four of the following variables, and the fifth is calculated, along with the duty:

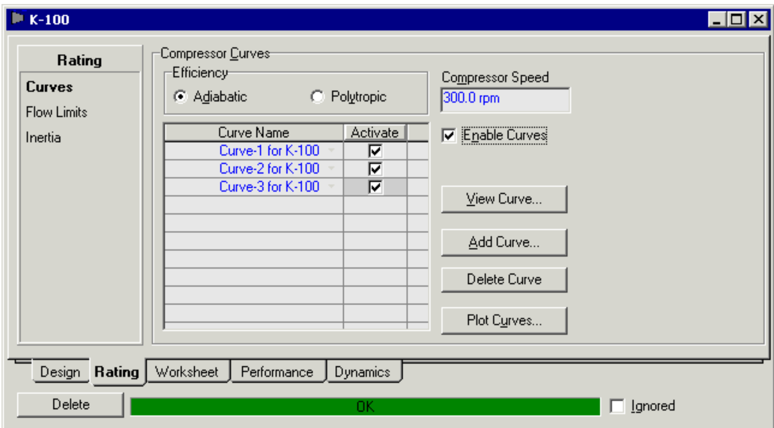
- Inlet Temperature
- Inlet Pressure
- Outlet Temperature
- Outlet Pressure
- Efficiency

It is assumed that you have specified the composition and flow.

Single MW (Molecular Weight)

If you choose the Single Curve radio button on the Parameters page of the Design Tab, the only group visible on the Curves page is the Compressor Curves group.

Figure 9.6



This option is only relevant for compressors and not expanders.

Entering Curve Data

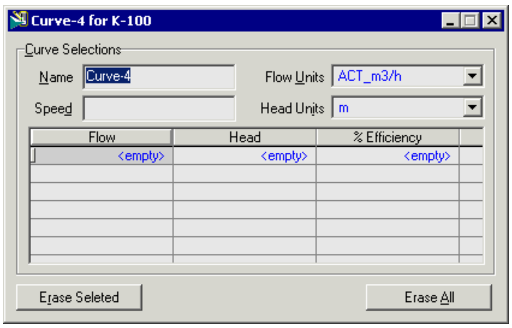
The following are steps to add a curve to the compressor or expander:

1. Select either the **Adiabatic** or **Polytropic** radio button in the Efficiency group. This determines the basis of your input efficiency values.

   The efficiency type must be the same for all input curves.

2. Click the **Add Curve** button and the Curve property view appears.

Figure 9.7



3. You can specify the following data in the Curve property view.

Curve Data	Description
Name	Name of the curve.
Speed	The rotational speed of the Centrifugal Compressor or Expander. This is optional if you specify only one curve. HYSYS can interpolate values for the efficiency and head of the Centrifugal Compressor or Expander for speeds that are not plotted.
Flow Units/Head Units	Units for the flow and head.
Flow/Head/% Efficiency	One row of data is equivalent to one point on the curve. For better results, you should enter data for at least three (or more) points on the curve.

4. Click the **Close** icon to return to the Curves page.
5. Select the corresponding **Activate** checkbox to use that curve in calculations.

6. For each additional curve, repeat steps #2 to #5.
7. Click the **Enable Curves** checkbox.

You can remove a specific curve from the calculation by clearing its **Activate** checkbox.

HYSYS uses the curve(s) to determine the appropriate efficiency for your operational conditions. If you specify curves, ensure the efficiency values on the Parameters page are empty or a consistency error will be generated.

Once a curve has been created, the following three buttons on the Curves page are enabled:

- **View Curve**. Allows you to view or edit your input data in the Curve property view.
- **Delete Curve**. Allows you to delete the selected curve from the simulation.
- **Plot Curves**. Allows you to view a graph of activated curves.

You can access the Curve property view of an existing curve by clicking the **View Curve** button or by double-clicking the curve name.

Deleting Curve Data

The following are two ways you can delete information within a curve:

1. Double-click the curve name to open the Curve property view.
2. Highlight the data you want to delete and click the **Erase Selected** button.

OR

1. Select the curve within the table and click the **View Curve** button.
2. Click the **Erase All** button to delete all of the information within the Curve property view.

Single Curve

When you have a single curve, the following combinations of input allow the operation to completely solve (assuming the feed composition and temperature are known):

- Inlet Pressure and Flow Rate
- Inlet Pressure and Duty
- Inlet Pressure and Outlet Pressure
- Inlet Pressure and Efficiency corresponding to the Curve type (for example, if the Curve is Adiabatic you need to provide an Adiabatic Efficiency).

Multiple Curves

If multiple curves have been installed, an operating speed is specified on the Curves page, and one of the multiple curves' speed equals the operating speed, then only the curve with the corresponding speed is used. For example, if you provide curves for two speeds (1000/min and 2000/min), and you specify an operating speed of 1000/min, then only the curve with the speed of 1000/min is used within the calculation.

If multiple curves have been installed, an operating speed has been specified, and none of the multiple curves' speed equals the operating speed, then all of the curves will be used within the calculation. For example, if you provide curves for two speeds (1000/min and 2000/min), and you specify an operating speed of 1500/min, HYSYS interpolates between the two curves to obtain the solution. You must also provide an inlet pressure and one of the following variables: flow rate, duty, outlet pressure, or efficiency, as explained above.

HYSYS can calculate the appropriate speed based on your input. In this case, you need to provide the feed composition, pressure, and temperature as well as two of the following four variables:

- Flow rate
- Duty
- Efficiency
- Outlet Pressure

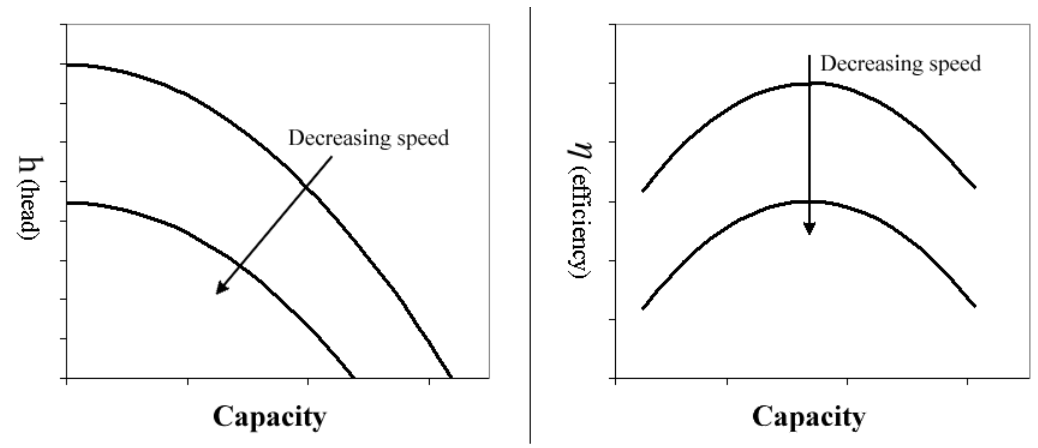
Once you provide the necessary information, the appropriate speed is determined, and the other two variables are then calculated.

Dynamics Mode

In order to run a stable and realistic dynamics model, HYSYS requires you to input reasonable curves. If compressors or expanders are linked, it is a good idea to ensure that the curves plotted for each unit operation span a common speed and capacity range.

Typical Compressor and Expander curves are plotted in **Figure 9.8** and **Figure 9.9**.

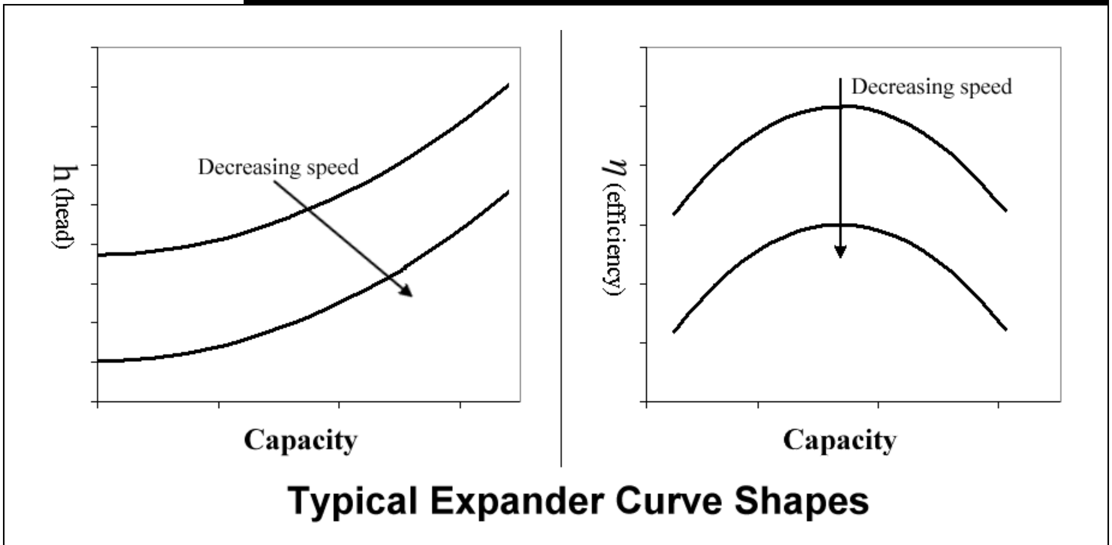
Figure 9.8



Typical Compressor Curve Shapes

For an Expander, the head is only zero when the speed and capacity are zero.

Figure 9.9



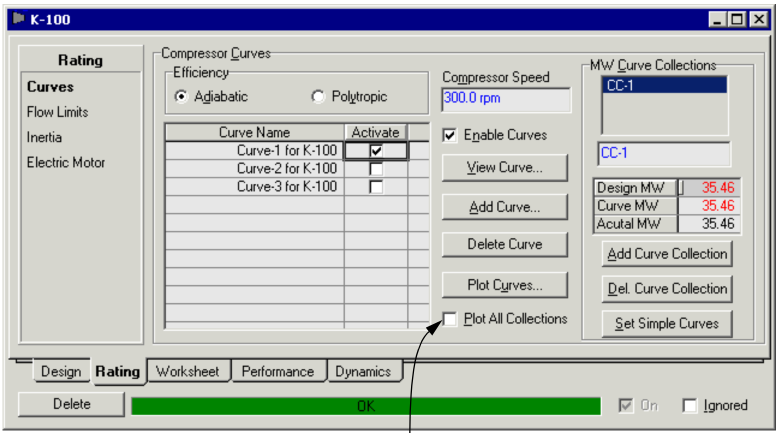
Multiple MW (Molecular Weight)

The Multiple MW (molecular weight) option is for more advanced users of HYSYS. This option allows the performance of the compressor to vary with the flowing gas molecular weight based upon the performance map you specified.

This option is only relevant for compressors and not expanders.

When you choose the Multiple MW Curves radio button on the Parameters page of the Design tab, the MW Curve Collections group and the **Plot All Collections** checkbox are added to the Curves page.

Figure 9.10



The **Plot All Collections** checkbox is useful if you want to see all the curves for all the curve collection (in the compressor) appear in one plot. This is particularly useful if all the curves data is based on one speed.

The following is a brief description of the fields and buttons found within the MW Curve Collections group.

Fields	Description
MW Curve Collections list	Displays the list of curve collection available in the unit operation. Each curve collection contains a set of data curves at a particular molecular weight.
Curve Collection name	Allows you to rename the selected curve collection in the list.
Design MW	The Design MW is the design average molecular weight for the compressor. Its default value is the same as the value for the Actual MW. This value is only used as a reference point and does not affect the Compressor calculation.

Fields	Description
Curve MW	Each curve collection has its corresponding set of curves at a particular molecular weight.
Actual MW	This value is calculated by HYSYS. The field displays the actual MW of the stream within your case. The following are descriptions of three potential operating situations: • If the Actual MW is the same as the Design MW, it means the compressor is operating with components at your designed MW. • If the Actual MW is less than the Design MW value, it means the compressor is operating with lighter components. • If the Actual MW is more than the Design MW value, it means the compressor is operating with heavier components.

Buttons	Description
Add Curve Collection	Adds another empty curve collection to the MW Curve Collections list.
Del. Curve Collection	Deletes the selected curve collection in the MW Curve Collections list.
Set Simple Curves	Creates two more curve collections with curve data based on the selected curve collection in the MW Curve Collections list. The data values within the two new curve collections are estimated values for testing purposes only. You should modify this data accordingly to specify your actual curve values.

Creating Multiple Curve Collections

The following are methods on how you can create multiple curve collections:

Refer to **Entering Curve Data** section for more information.

1. Enter the data for the curve(s). All of these values will be stored under a curve collection named **CurveCollection-1**.
2. Click the **Set Simple Curves** button.
3. Two hypothetical curve collections will appear (named **CurveCollection-2** and **CurveCollection-3**). These two new collections will provide you with rough data for curves generated with lighter or heavier components. These estimated values are based on the values entered for CurveCollection-1.

The values given in CurveCollection-2 and CurveCollection-3 are for testing purposes only. You need to modify the data accordingly in order to determine the definite curve values.

OR

Refer to **Entering Curve Data** section for more information.

1. Enter the data for the curve(s). All of these values will be stored under a curve collection.
2. Click the **Add Curve Collection** button.
3. Repeat step #1 to enter the data for the new curve collection.
4. Repeat steps #2 to #3 for each additional curve collection.

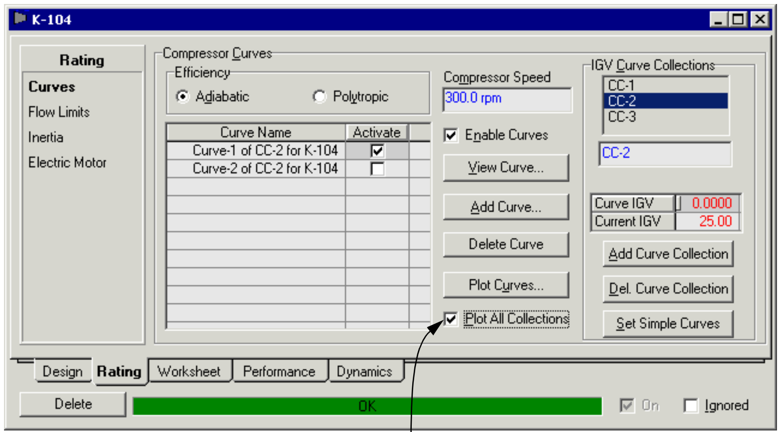
Multiple IGV (Inlet Guide Vane)

The Multiple IGV (inlet guide vane) Curves option allows you to model a compressor with adjustable guide vanes. The guide vanes are modulated to control the capacity of the compressor.

To use the Multiple IGV Curves feature, you need a manufacturers performance map of curves at different IGV operating conditions.

When you choose the Multiple IGV Curves radio button on the Parameters page of the Design tab, the IGV Curve Collections group and the **Plot All Collections** checkbox are added to the Curves page.

Figure 9.11



The **Plot All Collections** checkbox is useful if you want to see all the curves for all the curve collection (in the compressor) appear in one plot. This feature is particularly useful if all the curves data is based on one speed.

The following is a brief description of the fields and buttons found within the IGV Curve Collections group.

Fields	Description
IGV Curve Collections list	Displays the list of curve collection available in the unit operation. Each curve collection contains a set of data curves at a particular inlet guide vane.
Curve Collection name	Allows you to rename the selected curve collection in the list.
Curve IGV	Allows you to specify the inlet guide vane position that the entered curve set data is at (or for).
Current IGV	Allows you to specify the current position that the compressor is operating at. You can specify this value to different values during operation or specify this value in controllers during Dynamics mode.

Buttons	Description
Add Curve Collection	Adds another empty curve collection to the IGV Curve Collections list.

Buttons	Description
Del. Curve Collection	Deletes the selected curve collection in the IGV Curve Collections list.
Set Simple Curves	Creates two more curve collections with curve data based on the curve collection in the IGV Curve Collections list. The data values within the two new curve collections are estimated values for testing purposes only. You should modify this data accordingly to specify your actual curve values.

Creating Multiple Curve Collections

The following are methods on how you can create multiple curve collections:

Refer to **Entering Curve Data** section for more information.

1. Enter the data for the curve(s). All of these values will be stored under a curve collection named **CurveCollection-1**.
2. Click the **Set Simple Curves** button.
3. Two hypothetical curve collections will appear (named **CurveCollection-2** and **CurveCollection-3**). These two new collections will provide you with rough data for curves generated with lighter or heavier components. These estimated values are based on the values entered for CurveCollection-1.

The values given in CurveCollection-2 and CurveCollection-3 are for testing purposes only. You need to modify the data accordingly in order to determine the definite curve values.

OR

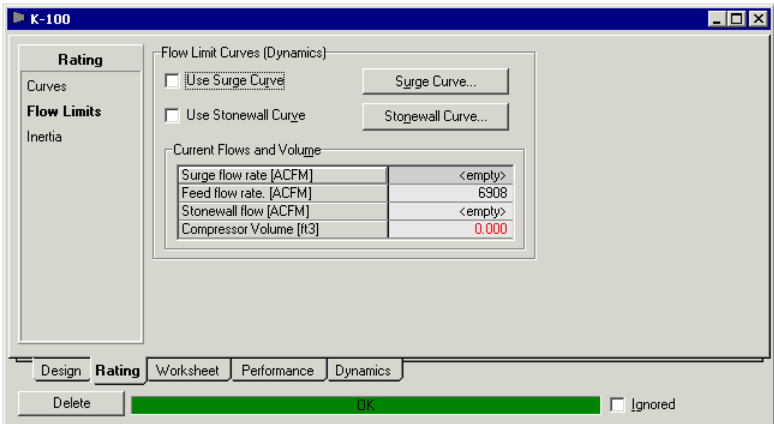
Refer to **Entering Curve Data** section for more information.

1. Enter the data for the curve(s). All of these values will be stored under a curve collection.
2. Click the **Add Curve Collection** button.
3. Repeat step #1 to enter the data for the new curve collection.
4. Repeat steps #2 to #3 for each additional curve collection.

Flow Limits Page

There is a certain range that the dynamic Centrifugal Compressors or Expanders can operate in depending on its operating speed. The lower flow limit of a Centrifugal Compressor is called the surge limit, whereas the upper flow limit is called the stonewall limit. In HYSYS, you can specify the flow limits of a Centrifugal Compressor or Expander by plotting surge and stonewall curves.

Figure 9.12



If you are working exclusively in Steady State mode, you are not required to change any information on the Flow Limits page.

From the Flow Limits page, it is possible to add Surge or Stonewall curves for the Centrifugal Compressor.

The procedure for adding or editing a Stonewall curve is similar to the procedure for adding or editing a Surge curve.

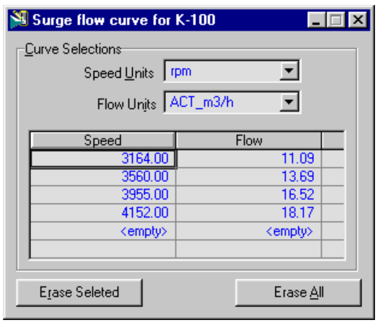
When a dynamic Centrifugal Compressor reaches its stonewall limit, HYSYS fixes the flow at that Centrifugal Compressor speed. When a Centrifugal Compressor reaches the surge limit, the flow reverses and cycles continuously causing damage to the Centrifugal Compressor. This phenomenon is modeled in HYSYS by causing the flow rate through the Centrifugal Compressor to fluctuate randomly below the surge flow.

Adding or Editing a Surge Curve

To add or edit a Surge curve, follow this procedure:

1. Click the **Surge Curve** button. The Surge flow curve property view appears.
2. From the Speed Units drop-down list, select the units you want to use for the speed measurements.
3. From the Flow Units drop-down list, select the units you want to use for the flow measurements.
4. Specify the speed and flow data points for the curve.
5. Once you have entered all the data points, click the **Close** icon to return to the Compressor or Expander property view.

Figure 9.13



6. Select the **Use Surge Curve** checkbox to use the surge curve for the compressor or expander calculations.

Deleting data of a Surge Curve

To delete data within a surge curve, do the following:

1. Click the **Surge Curve** button. The Surge flow curve property view appears.
2. Do one of the following:
    - To remove a certain data point, select either the speed cell or flow cell, and click the **Erase Selected** button.
    - To remove all the data points, click the **Erase All** button.

Nozzles Page

Refer to **Section 1.3.6 - Nozzles Page** for more information.

The Nozzles page contains information regarding the elevation and diameter of the nozzles.

If you are working exclusively in Steady State mode, you are not required to change any information on the Nozzles page.

For a Centrifugal Compressor or Expander unit operation it is strongly recommended that the elevation of the inlet and exit nozzles are equal. If you want to model static head, the entire piece of equipment can be moved by modifying the Base Elevation relative to Ground Elevation field.

Inertia Page

Refer to **Section 1.6.4 - Inertia** in the **HYSYS Dynamic Modeling** guide for more information.

The inertia modeling parameters and the friction loss associated with the impeller in the Centrifugal Compressor can be specified on the Inertia page.

If you are working exclusively in Steady State mode, you are not required to change any information on the Inertia page.

The HYSYS Dynamics license is required to use the Inertia features found on this page.

Electric Motor Page

The Electric Motor page allows you to drive your rotating unit operation through the designation of a motor torque versus speed curve. These torque vs. speed curves can either be obtained from the manufacturer for the electric motor being used or from a typical curve for the motor type.

For most process industry applications, a NEMA type A or B electric motor is used. When you use the Electric Motor option the torque (and power) generated by the motor is balanced against the torque consumed by the rotating equipment.

The Electric Motor functionality is only relevant in Dynamics mode.

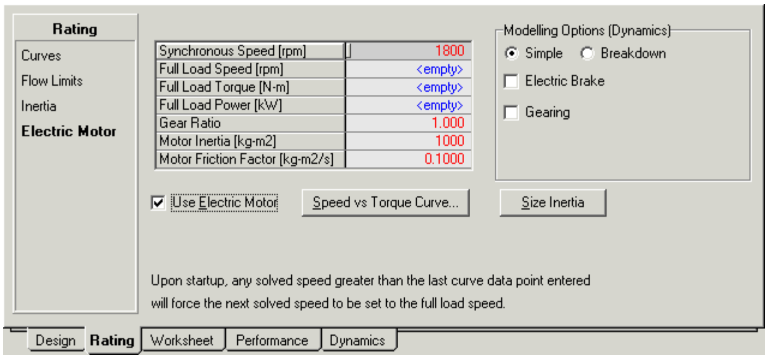
The Electric Motor option uses one degree of freedom in your dynamic specifications.

The results of the Electric Motor option are presented on the Power Page in the Performance Tab of the rotating equipment operation.

When you activate the Electric Motor option:

- An **On** checkbox will appear at the bottom of the Compressor property view, which can be used to turn the motor on and off.
- The Compressor operation icon in the PFD changes to include a motor.

Figure 9.14



The following table lists and describes the objects in the Electric Motor page:

Object	Description
Synchronous Speed cell	Allows you to specify the synchronous speed of the motor.
Full Load Speed cell	Allows you to specify the design speed of the motor.
Full Load Torque cell	Allows you to specify the design torque of the motor.
Full Load Power cell	Allows you to specify the design power of the motor.
Gear Ratio cell	Allows you to manipulate the gear ratio. The gear ratio is the rotating equipment's speed divided by the motor speed.
Motor Inertia cell	Allows you to specify the motor inertia.
Motor Friction Factor cell	Allows you to specify the motor friction factor.
User Electric Motor checkbox	Allows you to toggle between using or ignoring the electric motor functionality.
Speed vs Torque Curve button	Allows you to view the plot and specify the data in the **Speed vs. Torque Curve Property View**.
Size Inertia button	Allows you to calculate the inertia based on the following equation: $$I = 0.0043\left(\frac{P}{N}\right)^{1.48}$$ *where:* *I = inertia* $(kg \cdot m^3)$ *P = full load power of the motor (kW)* *N = full load speed of the motor (rpm/1000)*
Simple radio button	Allows you to select the **Simple** model for the modelling option.
Breakdown radio button	Allows you to select the **Breakdown** model for the modelling option.
Electric Brake checkbox	Allows you to model the torque force on the rotating equipment simply by changing the sign of the produced torque value.
Gearing checkbox	Allows the gear ratio to be updated during integration. A zero value for the gear ratio indicates a decoupling of the equipment.

Refer to **Operation Model** section for more information.

Theory

The definition of torque is found from the following equation:

$$T = \frac{P \times \omega \times 2 \times \pi}{1000 \times 60} \tag{9.11}$$

where:

P = power consumption (kW)

T = torque (Nm)

ω = synchronous speed (rpm)

The synchronous electric motor speed can be found from:

$$\omega = \frac{120f}{p} \tag{9.12}$$

where:

f = power supply frequency (Hz), typically either of 50 or 60

p = number of poles on the stator

The number of poles is always an even number of 2, 4, 6, 8, 10, and so forth. In North America, common motor speeds are always 3600, 1800, 1200, 900, 720, and so forth.

The relationships of inertia and friction loss in the total energy balance are the same as for the pump and compressor operations.

Operation Model

There are three ways to use the Electric Motor curve, each with progressing rigor.

- Simple Model. The easiest calculation is the Simple modelling option (default). This model is useful if you just want to model the startup/shutdown transient and want to keep the equipment at the fixed full load speed once operating. In this mode, once the speed has accelerated enough to become larger than the last (largest) curve speed value entered, the motor speed immediately is set to the full load speed and remains there until the motor is turned off. If the process invokes a larger torque than the motor curve suggests the motor can produce, the speed still remains synchronous and remains at its full load value.

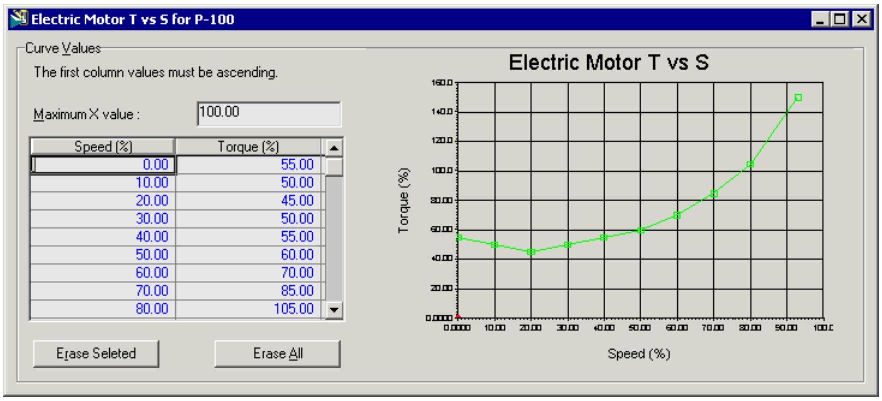
- Breakdown Model. The Breakdown modelling option allows the speed to reduce if the system torque or resistance gets too large. To use this option, the largest (last) curve speed value entered should be just less than the full load speed value. This provides for a smooth transition in operation. You can drop the curve to a lower torque than the breakdown torque if desired.
- Simple Model Modified. The third modelling approach is to use the Simple model option, but enter the speed vs torque curve up to a speed value of 99.99% of the synchronous speed. In this case the full load speed entered is only used, if necessary, to calculate the full load torque and is not used otherwise. With this approach, the speed vs torque curve must ascend or drop from the breakdown torque to approach zero torque at 100% speed. For most motor types, this approach is nearly vertical (asymptotic). This modelling approach allows for the simulation of the slippage of the motor speed based upon the actual and current system resistance. The operating speed of the motor will then move based upon the process model operation. Use a near vertical curve to keep a constant speed or level it off more to allow greater slip. This performance should be predicted by using an accurate manufacturers torque vs speed curve.

The speed and torque are not solved simultaneously with the pressure flow solution but instead is lagged by a time step. You may need to use a smaller time step to ensure accuracy and pressure flow solver convergence.

Speed vs. Torque Curve Property View

The Speed vs. Torque Curve property view displays the data curve of speed versus torque in both table and plot format.

Figure 9.15



To access the Speed vs. Torque Curve property view, click the **Speed vs Torque Curve** button on the **Electric Motor** page of the **Rating** tab.

The values under the Speed and Torque columns are entered as a percent of the Full Load values.

The Speed vs. Torque curve must always contain a 0% speed value.

The maximum table speed cannot be greater than the value in the Maximum X value field. The Maximum X value is the ratio of the full load speed to the synchronous speed.

During integration, the current operating point appears on the Torque vs. Speed curve.

The following table lists and describes the objects available in the Speed vs. Torque Curve property view:

Object	Description
Maximum X value display field	Displays the ratio of the full load speed to the synchronous speed.
Speed column	Allows you to specify speed percentage values you want to plot.

Object	Description
Torque column	Allows you to specify the torque percentage values associated with the speed.
Erase Selected button	Allows you to delete the row containing both speed and torque percentage values of the selected cell.
Erase All button	Allows you to delete all the values in the table.

9.1.5 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the Centrifugal Compressor or Expander.

The PF Specs page is relevant to dynamics cases only.

9.1.6 Performance Tab

The Performance page contains the calculated results of the compressor or expander.

Results Page

On the Results page, you can view a table of calculated values for the Centrifugal Compressor or Expander:

- Adiabatic Head
- Polytropic Head
- Adiabatic Fluid Head
- Polytropic Fluid Head
- Adiabatic Efficiency
- Polytropic Efficiency
- Power Produced
- Power Consumed
- Friction Loss
- Rotational inertia
- Fluid Power
- Polytropic Head Factor
- Polytropic Exponent
- Isentropic Exponent
- Speed

Power Page

The Power page is only available for the compressor. The information displayed in this page is:

- Compressor rotor power
- Compressor rotor torque
- Electric motor power
- Electric motor torque
- Electric motor speed

9.1.7 Dynamics Tab

The Dynamics tab contains the following pages:

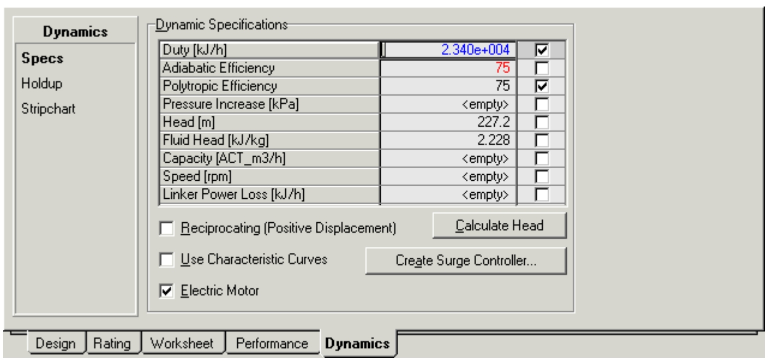
- Specs
- Holdup
- Stripchart

If you are working exclusively in Steady State mode, you are not required to change any information on the pages accessible through this tab.

Specs Page

The dynamic specifications of the Centrifugal Compressor or Expander can be specified on the Specs page.

Figure 9.16



In general, two specifications are required in the Dynamics Specifications group. You should be aware of specifications which may cause complications or singularity in the pressure flow matrix. Some examples of such cases are:

- The **Pressure Increase** checkbox should not be selected if the inlet and exit stream pressures are specified.
- The **Speed** checkbox should not be selected if the **Use Characteristic Curves** checkbox is not selected.

The possible dynamic specifications are as follows:

Duty

The duty is defined, in the case of the Centrifugal Compressor operation, as the power required to rotate the shaft and provide energy to the fluid. The duty has three components:

Duty = Power imparted to the fluid + Power required to change the rotational speed of the shaft + Power lost due to mechanical friction loss (9.13)

The duty in a Centrifugal Compressor should be specified only if there is a fixed power available to be used to drive the shaft.

Efficiency (Adiabatic and Polytropic)

For a dynamic Centrifugal Compressor, the efficiency is given as the ratio of the isentropic power required for compression to the actual energy imparted to the fluid. The efficiency, η, is defined as:

$$\eta = \frac{W(to\ system)}{F_1(MW)(h_2 - h_1)} \tag{9.14}$$

where:

W = *isentropic power*

F_1 = *molar flow rate of the inlet gas stream*

MW = *molecular weight of the gas*

h_1 = *inlet head*

h_2 = *outlet head*

For a dynamic Expander, the efficiency, η, is defined as:

$$\eta = \frac{F_1(MW)(h_1 - h_2)}{W(from\ system)} \tag{9.15}$$

If a polytropic efficiency definition is required, the polytropic work should be provided in **Equation (9.14)** or **Equation (9.15)**. If an adiabatic efficiency definition is required, the isentropic work should be provided.

The general definition of the efficiency does not include the losses due to the rotational acceleration of the shaft and seal losses. Therefore, the efficiency equations in dynamics are not different from the general efficiency equations defined in **Section 9.1.1 - Theory**. This is true since the actual work required by a steady state Centrifugal Compressor is the same as the energy imparted to the fluid.

If the Centrifugal Compressor or Expander curves are specified in the Curves page of the Rating tab, the adiabatic or polytropic efficiency can be interpolated from the flow of gas and the speed of the Centrifugal Compressor or Expander.

Pressure Increase

A Pressure Increase specification can be selected if the pressure drop across the Centrifugal Compressor is constant.

Head

The isentropic or polytropic head, h, can be defined as a function of the isentropic or polytropic work. The relationship is:

$$W = (MW)F_1(CF)gh \tag{9.16}$$

where:

W = isentropic or polytropic power

MW = molecular weight of the gas

CF = correction factor

F_1 = molar flow rate of the inlet gas stream

g = gravity acceleration

If the Centrifugal Compressor or Expander curves are provided in the Curves page of the Rating tab, the isentropic or polytropic head can be interpolated from the flow of gas and the speed of the Centrifugal Compressor or Expander.

Fluid Head

The Fluid Head is the produced head in units of energy per unit mass.

Capacity

The capacity is defined as the actual volumetric flow rate entering the Centrifugal Compressor or Expander. A capacity specification can be selected if the volumetric flow to the unit operation is constant.

Speed

The rotational speed of the shaft, ω, driving the Centrifugal Compressor or being driven by the Expander can be specified.

Shift to Reciprocating Compressor (Positive Displacement)

Refer to **Section 9.2 - Reciprocating Compressor** for more information.

Select the **Reciprocating (Positive Displacement)** checkbox if you want to change the Centrifugal Compressor to a Reciprocating Compressor. You can change the Centrifugal Compressor to a Reciprocating Compressor at any time.

The reciprocating checkbox option is only available with the compressor unit operation.

Use Characteristic Curves

Select the **Use Characteristic Curves** checkbox, if you want to use the curve(s) specified in the Curves page of the Rating tab. If a single curve is specified in a dynamics Centrifugal Compressor, the speed of the Centrifugal Compressor is not automatically set to the speed of the curve (unlike the steady state Centrifugal Compressor or Expander unit operation). A different speed can be specified and HYSYS extrapolates values for head and efficiency.

Linker Power Loss

Select the **Linker Power Loss** checkbox, if you want to specify the power loss (negative for a power gain) of the linked operations.

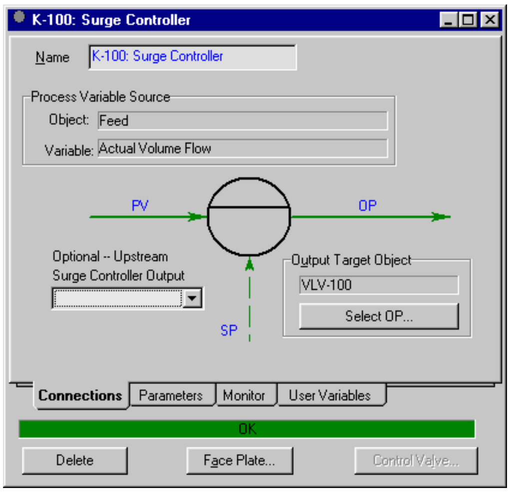
Electric Motor

Select the **Electric Motor** checkbox if you want to use the electric motor functionality.

Surge Controller

The Create Surge Controller button on the Specs page of the Dynamics tab opens a Surge Controller property view (which is owned by the Centrifugal Compressor). If you decide to delete the Centrifugal Compressor, the surge controller associated with the Centrifugal Compressor is deleted as well. The surge controller also works exclusively with Centrifugal Compressor and Expander unit operations.

Figure 9.17



As mentioned, a Centrifugal Compressor surges if its capacity falls below the surge limit. The surge controller determines a minimum volumetric flow rate that the Centrifugal Compressor should operate at without surging. This is called the surge flow. The surge controller then attempts to control the flow to the Centrifugal Compressor at some percent above the surge flow (this is typically 10%). The surge controller essentially acts like PID Controller operations. The control algorithms used to prevent Centrifugal Compressors from surging are extensions of the PID algorithm.

There are two major differences which distinguish a surge controller and a regular controller:

- The setpoint of the surge controller is calculated and not set.
- More aggressive action is taken by the surge controller if the Centrifugal Compressor is close to surging.

Connections Tab

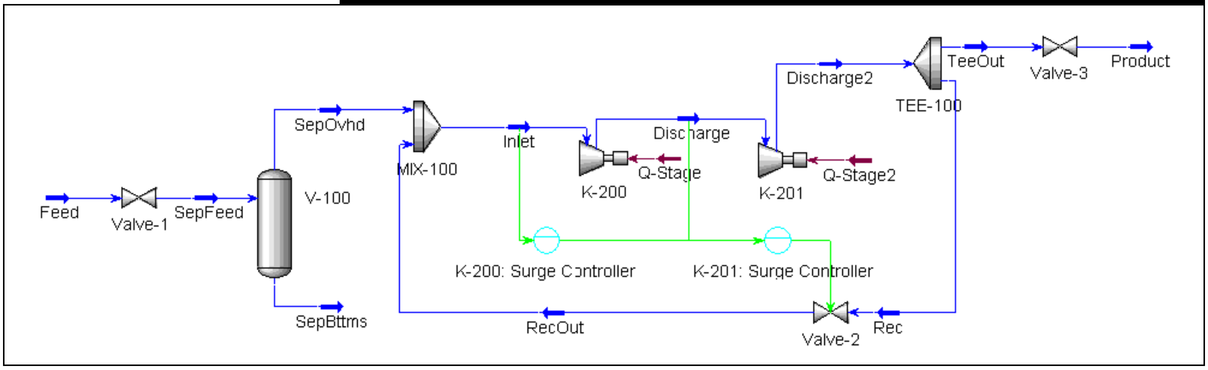
For more information on the individual parameters which make up the Connections tab, refer to **Chapter 5 - Logical Operations**

The Connections tab is very similar to a PID controller's Connections tab. The inlet volumetric flow to the Centrifugal Compressor is automatically defaulted as the process variable (PV) to be measured. You must select a Control Valve operation as an operating variable (OP), which has a direct effect on the inlet flow to the Centrifugal Compressor.

The Upstream Surge Controller Output field contains a list of the other surge controllers in the simulation flowsheet. If you select an upstream surge controller using the Upstream Surge Controller Output field, HYSYS ensures that the output signal of the Centrifugal Compressor's surge controller is not lower than an upstream surge controller's output signal.

Consider a situation in which two compressors are connected in series.

Figure 9.18



As shown in the figure above, both surge controllers must use the same valve for surge control. If the surge controllers are connected in this manner HYSYS autoselects the largest controller output. This is done to ensure that surge control is adequately provided for both compressors.

Parameters Tab

The parameters tab consists of the following pages:

- Configuration
- Surge Control

Configuration Page

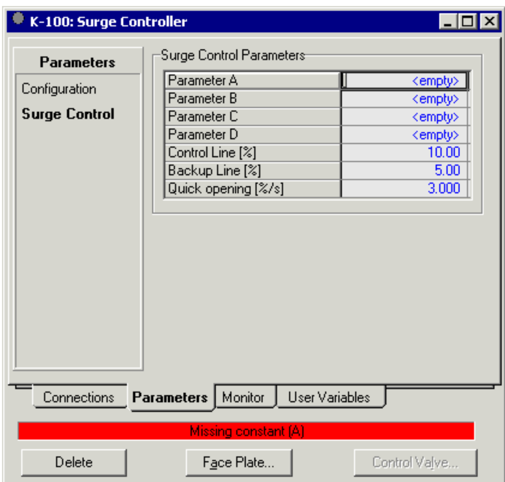
For more information on the individual fields in the Configuration page, refer to **Section 5.4.4 - PID Controller**.

If the process variable (PV) is operating above a certain margin over the surge flow limit, the surge controller operates exactly as a PID Controller. Therefore, PID control parameters should be set on the Configuration page. The process variable range, the controller action, operation mode, and the tuning parameters of the controller can be set in this page.

Surge Control Page

Various surge control parameters can be specified on the Surge Control page.

Figure 9.19



A head versus quadratic flow expression relates the surge flow to the head of the Centrifugal Compressor.

$$h_m = A + B(F_s) + C(F_s)^2 + D(F_s)^3 \tag{9.17}$$

where:

F_s = *surge flow* (m^3/s)

h_m = *head of the Centrifugal Compressor*

A, B, C, D = parameters used to characterize the relationship between surge flow and head

You can enter surge flow parameters *A*, *B*, *C*, and *D* in order to characterize the relationship between the surge flow and head.

The next three parameters in the Surge Control Parameters section are defined as follows:

Surge Control Parameter	Description
Control Line (%)	The primary setpoint for the surge controller. This line is defaulted at 10% above the surge flow. If the flow is above the backup line then the surge controller acts as a normal PID controller.
Backup Line (%)	Set somewhere between the control line and the surge flow. This line is defaulted at 5% above the surge flow. If the flow to the Centrifugal Compressor falls below the backup line, more aggressive action is taken by the controller to prevent a surge condition.
Quick Opening (%/sec)	Aggressive action is taken by increasing the desired actuator opening at a rate specified in this field until the volumetric flow to the Centrifugal Compressor rises above the backup line.

Monitor Tab

The Monitor tab displays a chart that graphs the three variables (PV, SP, and OP) of the surge controller

User Variables Tab

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables tab enables you to create and implement your own user variables for the current operation.

Holdup Page

Refer to **Section 1.3.3 - Holdup Page** for more information.

Typical Centrifugal Compressors and Expanders in actual plants usually have significantly less holdup than most other unit operations in a plant. Therefore, the volume of the Centrifugal Compressor or Expander operation in HYSYS cannot be specified and is assumed to be zero on the Holdup page.

Stripchart Page

Refer to **Section 1.3.7 - Stripchart Page / Tab** for more information.

The Stripchart page allows you to select and create default strip charts containing various variable associated to the operation.

9.2 Reciprocating Compressor

In **Section 9.1 - Centrifugal Compressor or Expander** a Centrifugal Compressor type is presented. The following section discusses a Reciprocating Compressor. A Reciprocating Compressor is just another type of compressor used for applications where higher discharge pressures and lower flows are needed. It is known as a positive displacement type. Reciprocating Compressors have a constant volume and variable head characteristics, as compared to the Centrifugal Compressor that has a constant head and variable volume.

For Reciprocating Compressors there is no direct relationship between the head and flow capacity.

In HYSYS, Centrifugal and Reciprocating Compressors are accessed via the same compressor unit operation. However, the solution methods differ slightly as a Reciprocating Compressor does not require a compressor curve and the required geometry data. The present capability of Reciprocating Compressors in HYSYS is focused on a single stage compressor with a single or double acting piston. A typical solution method for a Reciprocating Compressor is as follows:

- Always start with a fully defined inlet stream, in other words, inlet pressure, temperature, flow rate, and compositional data are known.
- Specify compressor geometry data, for example, number of cylinders, cylinder type, bore, stroke, and piston rod diameter. HYSYS provides default values too.
- Compressor performance data, in other words, adiabatic efficiency or polytropic efficiency, and constant volumetric efficiency loss are specified.
- HYSYS calculates the duty required, outlet temperature if the outlet pressure is specified.

Some of the features in the dynamic Reciprocating Compressor unit operation include:

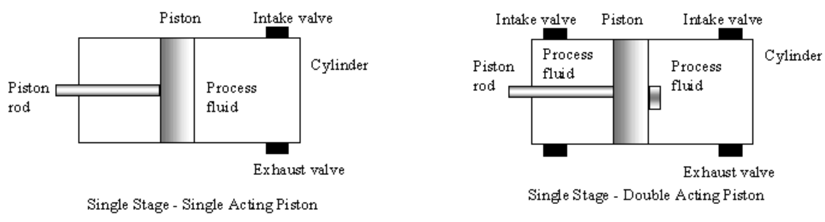
- Dynamic modeling of friction loss and inertia.
- Dynamic modeling which supports shutdown and startup behaviour.
- Dynamic modeling of the cylinder loading.
- Linking capabilities with other rotational equipment operating at the same speed with one total power.

9.2.1 Theory

In a single stage Reciprocating Compressor, it comprises of the basic components like the piston, the cylinder, head, connecting rod, crankshaft, intake valve, and exhaust valve. This is illustrated in **Figure 9.20**. HYSYS is capable of modeling a multi-cylinder in one Reciprocating Compressor with a single acting or double acting piston.

A single acting compressor has a piston that is compressing the gas contained in the cylinder using one end of the piston only. A double acting compressor has a piston that is compressing the gas contained in the cylinder using both ends of the piston. The piston end that is close to the crank is called crank end, while the other is named as outer.

Figure 9.20



The thermodynamic calculations for a Reciprocating Compressor are the same as a Centrifugal Compressor. Basically, there are two types of compression being considered:

- **Isentropic/adiabatic reversible path**. A process during which there is no heat added to or removed from the system, and the entropy remains constant. PV^k=constant, where k is the ratio of the specific heat (Cp/Cv).
- **Polytropic reversible path**. A process in which changes in the gas characteristic during compression are considered.

Details of the equation are found in **Section 9.1.1 - Theory**. Reference[1] has the information about the operation of the Reciprocating Compressor.

The performance of the Reciprocating Compressor is evaluated based on the volumetric efficiency, cylinder clearance, brake power, and duty.

Cylinder clearance, *C*, is given as:

$$C = \frac{\text{Sum of all clearance volume for all cylinders}}{PD} \tag{9.18}$$

where:

PD = positive displacement volume

The sum of all clearance volume for all cylinders includes both fixed and variable volume. *C* is normally expressed in a fractional or percentage form.

The piston displacement, *PD,* is equal to the net piston area multiplied by the length of piston sweep in a given period of time. This displacement can be expressed as follows:

- For a single-acting piston compressing on the outer end only:

$$PD = \frac{\pi \cdot D^2 \cdot \text{stroke}}{4} \tag{9.19}$$

- For a single-acting piston compressing on the crank end only:

$$PD = \frac{\pi \cdot (D^2 - d^2) \cdot \text{stroke}}{4} \tag{9.20}$$

- For double-acting piston (other than tail rod type):

$$PD = \frac{\pi \cdot (2D^2 - d^2) \cdot \text{stroke}}{4} \tag{9.21}$$

- For a double-acting piston (tail rod type):

$$PD = \frac{\pi \cdot (2D^2 - 2d^2) \cdot \text{stroke}}{4} \tag{9.22}$$

where:

d = piston rod diameter

D = piston diameter

PD includes the contributions from all cylinders and both ends of any double acting. If a cylinder is unloaded then its contribution does not factor in.

The volumetric efficiency is one of the important parameters used to evaluate the Reciprocating Compressor's performance. Volumetric efficiency, *VE*, is defined as the actual pumping capacity of a cylinder compared to the piston displacement volume.

VE is given by:

$$VE = \left[(1-L) - C\left[\frac{Z_s}{Z_d}\left(\frac{P_d}{P_s}\right)^{\frac{1}{k}} - 1\right]\right] \tag{9.23}$$

where:

P_d = *discharge pressure*

P_s = *suction pressure*

L = *effects of variable such as internal leakage, gas friction, pressure drop through valves, and inlet gas preheating*

k = *heat capacity ratio, Cp/Cv*

Z_d = *discharge compressibility factor*

Z_s = *suction compressibility factor*

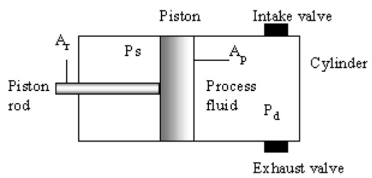
C = *clearance volume*

To account for losses at the suction and discharge valve, an arbitrary value about 4% VE loss is acceptable. For a non-lubricated compressor, an additional 5% loss is required to account for slippage of gas. If the compressor is in propane, or similar heavy gas service, an additional 4% should be subtracted from the volumetric efficiency. These deductions for non-lubricated and propane performance are both approximate, and if both apply, cumulative. Thus, the value of *L* varies from (0.04 to 0.15 or more) in general.

Rod Loading

Rod loads are established to limit the static and inertial loads on the crankshaft, connecting rod, frame, piston rod, bolting, and projected bearing surfaces.

Figure 9.21



It can be calculated as follows:

Load in compression, L_c

$$L_c = P_d A_p - P_s(A_p - A_r) \quad (9.24)$$

Load in tension, L_t

$$L_t = P_d(A_p - A_r) - P_s A_p \quad (9.25)$$

Maximum Pressure

The maximum pressure that the Reciprocating Compressor can achieve is:

$$P_{max} = P_s \cdot PR_{max} \quad (9.26)$$

Where the maximum discharge pressure ratio, PR_{max}, is calculated from:

$$PR_{max} = \left[\frac{Z_d}{Z_s \cdot C}(1 - L - VE + C)\right]^k \quad (9.27)$$

Flow

Flow into the Reciprocating Compressor is governed by the speed of the compressor. If the speed of the compressor is larger than zero then the flow rate is zero or larger then zero (but never negative). The molar flow is then equal to:

$$F = \left[\left(1 - \frac{L}{100}\right) - C\left[\frac{Z_s}{Z_d}\left(\frac{P_d}{P_s}\right)^{\frac{1}{k}} - 1\right]\right]\left[\frac{\frac{N}{60} \cdot PD \cdot \rho}{MW}\right] \tag{9.28}$$

where:

N = speed, rpm

ρ *= gas density*

MW = gas molecular weight

If the speed of the compressor is exactly zero, then the flow through the unit is governed by a typical pressure flow relationship, and you can specify the resistance in zero speed flow resistance, $k_{\text{zero speed}}$.

The flow equation is as follows:

$$\mathrm{F} = k_{\text{zero speed}} \cdot \sqrt{\rho \cdot \Delta P_{\text{friction}}} \tag{9.29}$$

where:

$\Delta P_{\text{friction}}$ *= frictional pressure drop across the compressor*

9.2.2 Reciprocating Compressor Property View

There are two ways that you can add a Compressor to your simulation:

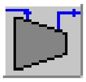
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Rotating Equipment** radio button.
3. From the list of available unit operations, select **Compressor**.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

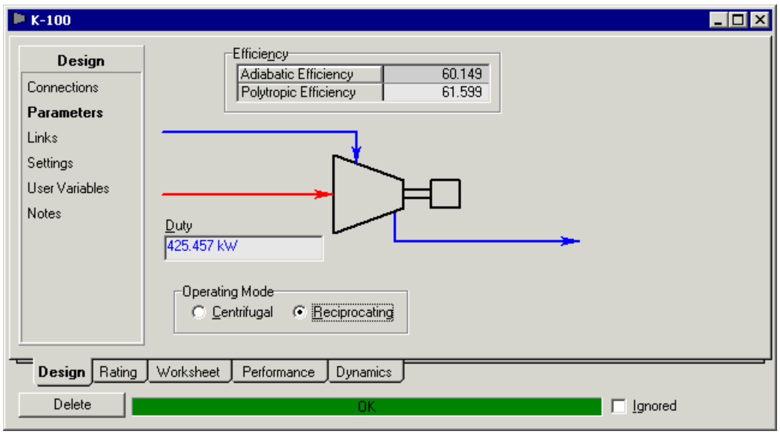
   You can also open the Object Palette by pressing **F4**.
2. Double-click the **Compressor** icon.



Compressor icon

The Compressor property view appears.

Figure 9.22



Do one of the following to complete the Reciprocating Compressor installation:

- On the **Design** tab, click the **Parameters** page. Select the **Reciprocating** radio button in the Operating Mode group.
- On the **Dynamics** tab, click on the **Specs** page. Select the **Reciprocating (Positive Displacement)** checkbox in the Dynamic Specifications group.

9.2.3 Design Tab

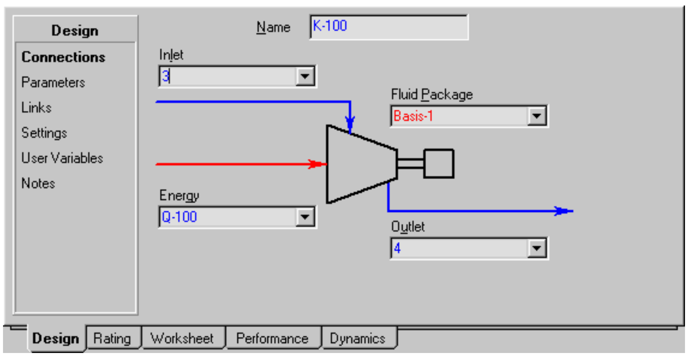
The Design tab contains the following pages:

- Connections
- Parameters
- Links
- Settings
- User Variables
- Notes

Connections Page

The Connections page allows you to specify the inlet stream, outlet stream, and energy stream.

Figure 9.23



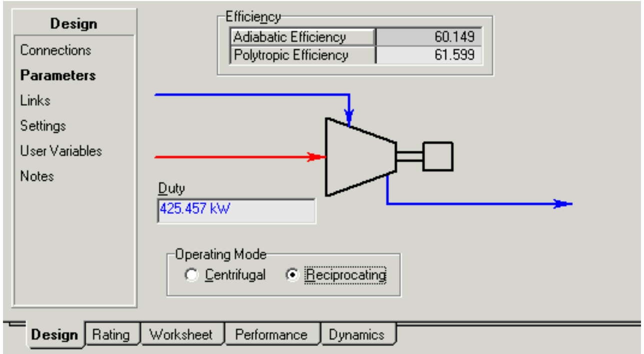
Refer to the section on the Centrifugal Compressor or Expanders **Connections Page** for more information.

The Connections page is identical to the Connections page for the Centrifugal Compressor property view.

Parameters Page

The Parameters pages is identical to the Centrifugal Compressor as shown in the figure below.

Figure 9.24



The difference between the Centrifugal and Reciprocating compressor is the missing Curve Input Option group.

You can switch between Centrifugal and Reciprocating Compressor by selecting one of the radio buttons in the Operating Mode group.

You can specify the duty of the attached energy stream on this page, or allow HYSYS to calculate it. The adiabatic and polytropic efficiencies appear as well.

You can specify only one efficiency, either adiabatic or polytropic. If you specify one efficiency and a solution is obtained, HYSYS back calculates the other efficiency, using the calculated duty and stream conditions.

The Reciprocating Compressor has a higher adiabatic efficiency than the Centrifugal Compressor, normally in the range of 85% - 95%.

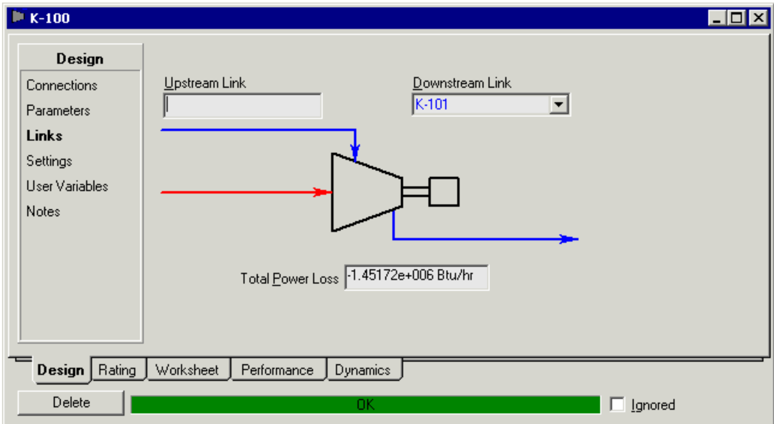
Maximum pressure ratio can be achieved at zero volume efficiency.

Links Page

Refer to **Links Page** section for more information.

The Links page is identical to the Centrifugal Compressor as shown in the figure below.

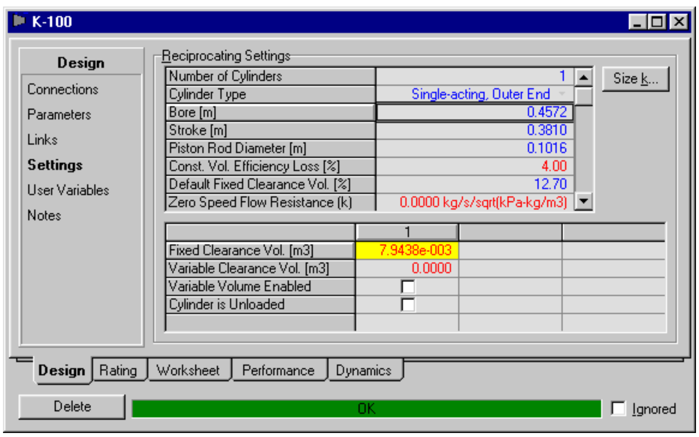
Figure 9.25



Settings Page

The Settings page is used to size the Reciprocating Compressor.

Figure 9.26



The Settings page is only visible when you have activated the Reciprocating Compressor option either from the Parameters page on the Design tab or the Specs page on the Dynamics tab.

A Reciprocating Compressor does not require a characteristic curve, however the following compressor geometry information is required:

- Number of Cylinders
- Cylinder Type
- Bore

Bore is the diameter of the cylinder.

- Stroke

Stroke is the distance head of piston travels.

- Piston Rod Diameter
- Constant Volumetric Efficiency Loss
- Default Fixed Clearance Volume
- Zero Speed Flow Resistance (k) - dynamics only
- Typical Design Speed

Typical Design Speed is the estimated speed for the rotor.

- Volumetric Efficiency
- Speed

Speed is the actual speed of the rotor.

Depending on the cylinder type selected, you have four parameters that can be specified. If the cylinder type is of double action, you need to specify the fixed clearance volume for the crank side and the outer side.

- Fixed Clearance Volume
- Variable Clearance Volume
- Variable Volume Enabled
- Cylinder is Unloaded - dynamics only

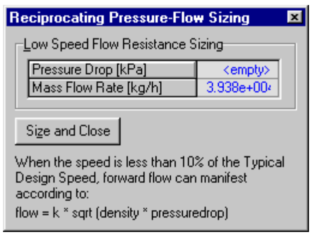
If the **Variable Volume Enabled** checkbox is selected, you need to specify a variable clearance volume.

The variable clearance volume is used when additional clearance volume (external) is intentionally added to reduce cylinder capacity.

If the **Cylinder is Unloaded** checkbox is selected, the total displacement volume is not considered and is essentially zero.

The **Size k** button allows you to access the Reciprocating Pressure-Flow Sizing property view, and specify a pressure drop and mass flow rate that is used to calculate the zero speed flow resistance of the Reciprocating Compressor.

Figure 9.27



User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

9.2.4 Rating Tab

The Rating tab contains the following pages:

- Nozzles
- Inertia
- Electric Motor

Nozzles Page

Refer to the section on the **Nozzles Page** for more information.

If you are working exclusively in Steady State mode, you are not required to change any information on the Nozzles page. The Nozzles page in the Reciprocating Compressor is identical to the Nozzles page in the Centrifugal Compressor.

Inertia Page

Refer to the section on the **Inertia Page** for more information.

If you are working exclusively in Steady State mode, you are not required to change any information on this page. The Reciprocating Compressor Inertia page is identical to the one for the Centrifugal Compressor.

Electric Motor

Refer to the section on the **Electric Motor Page** for more information.

If you are working exclusively in Steady State mode, you are not required to change any information on this page. The Reciprocating Compressor Electric Motor page is identical to the one for the Centrifugal Compressor.

9.2.5 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

The PF Specs page is relevant to dynamics cases only.

9.2.6 Performance Tab

The Performance tab consists of the Results page.

Results Page

On the Results page, you can view a table of calculated values for the Compressor.

In the Results group you will find the following fields:

- Adiabatic Head
- Polytropic Head
- Adiabatic Efficiency
- Polytropic Efficiency
- Power Consumed
- Friction Loss
- Rational Inertia
- Fluid Power
- Polytropic Head Factor
- Polytropic Exponent
- Isentropic Exponent
- Speed

In the Reciprocating group you will find the following fields:

- Total Effective Piston Displacement Volume
- Total Effective Fractional Clearance Volume
- Maximum Pressure Ratio
- Load in Compression
- Load in Tension

9.2.7 Dynamics Tab

Refer to **Section 9.1.7 - Dynamics Tab** for more information.

The Dynamics tab is identical to the one for the Centrifugal Compressor. However, when using a Reciprocating Compressor you cannot use the Characteristic Curves specification or create a Surge Controller.

If you are working exclusively in Steady State mode, you are not required to change any information on the pages accessible through the Dynamics tab.

9.3 Pump

The Pump operation is used to increase the pressure of an inlet liquid stream. Depending on the information specified, the Pump calculates either an unknown pressure, temperature or pump efficiency.

The dynamics Pump operation is similar to the Compressor operation in that it increases the pressure of its inlet stream. The Pump operation assumes that the inlet fluid is incompressible.

Some of the features in the dynamic Pump include:

- Dynamic modeling of friction loss and inertia.
- Dynamic modeling which supports shutdown and startup behaviour.
- Multiple head and efficiency curves.
- Modeling of cavitation if Net Positive Suction Head (NPSH) is less than a calculated NPSH limit.
- Linking capabilities with other rotational equipment operating at the same speed with one total power.

9.3.1 Theory

Calculations are based on the standard pump equation for power, which uses the pressure rise, the liquid flow rate, and density:

$$Power\ Required_{ideal} = \frac{(P_{out} - P_{in}) \times Flow\ Rate}{Liquid\ Density} \tag{9.30}$$

where:

P_{out} = *pump outlet pressure*

P_{in} = *pump inlet pressure*

The previous equation defines the ideal power needed to raise the liquid pressure.

The actual power requirement of the Pump is defined in terms of the Pump Efficiency:

$$Efficiency(\%) = \frac{Power\ Required_{ideal}}{Power\ Required_{actual}} \times 100\% \qquad (9.31)$$

When the efficiency is less than 100%, the excess energy goes into raising the temperature of the outlet stream.

Combining the above equations leads to the following expression for the actual power requirement of the Pump:

$$Power\ Required_{actual} = \frac{(P_{out} - P_{in}) \times Flow\ Rate \times 100\%}{Liquid\ Density \times Efficiency(\%)} \qquad (9.32)$$

Finally, the actual power is equal to the difference in heat flow between the outlet and inlet streams:

$$Power\ Required_{\text{actual}} = (Heat\ Flow_{\text{outlet}} - Heat\ Flow_{\text{inlet}}) \qquad (9.33)$$

If the feed is fully defined, only two of the following variables need to be specified for the Pump to calculate all unknowns:

- Outlet Pressure or Pressure Drop
- Efficiency
- Pump Energy

HYSYS can also back-calculate the inlet Pressure.

> **For a pump, an efficiency of 100% does not correspond to a true isentropic compression of the liquid. Pump calculations are performed by HYSYS with the assumption that the liquid is incompressible; that is, the density is constant (liquid volume is independent of pressure). This is the usual assumption for liquids well removed from the critical point, and the standard pump equation given above is generally accepted for calculating the power requirement. However, if you want to perform a more rigorous calculation for pumping a compressible liquid (for example, one near the critical point), you should instead install a compressor to represent the pump.**

If you choose to represent a Pump by installing a Compressor in HYSYS, the power requirement and temperature rise of the Compressor is always greater than those of the Pump (for the same fluid stream), because the compressor treats the liquid as a compressible fluid. When the pressure of a compressible fluid increases, the temperature also increases, and the specific volume decreases. More work is required to move the fluid than if it were incompressible, exhibiting little temperature rise, as is the case with a HYSYS Pump.

The ideal power required, W, to increase the pressure of an incompressible fluid is:

$$W = \frac{(P_2 - P_1)F(MW)}{\rho} \tag{9.34}$$

where:

P_1 = *pressure of the inlet stream*

P_2 = *pressure of the exit stream*

ρ = *density of the inlet stream*

F = *molar flow rate of the stream*

MW = *molecular weight of the fluid*

For a Pump, it is assumed that the entering liquid stream is incompressible. Therefore, the ideal work defined in **Equation (9.34)** does not correspond to a true isentropic compression of the liquid. Despite this, the pump efficiency is defined in terms of the ideal work and not the isentropic work.

Incompressibility is the usual assumption for liquids well removed from the critical point, and the standard pump equation provided in **Equation (9.34)** is generally accepted for calculating the power requirement. However, if you want to perform a more rigorous calculation for pumping a compressible liquid (for example, one near the critical point), you should install a Compressor operation instead of a Pump.

9.3.2 Pump Property View

There are two ways that you can add a Pump to your simulation:

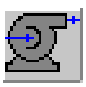
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Rotating Equipment** radio button.
3. From the list of available unit operations, select **Pump**.
4. Click the **Add** button.

OR

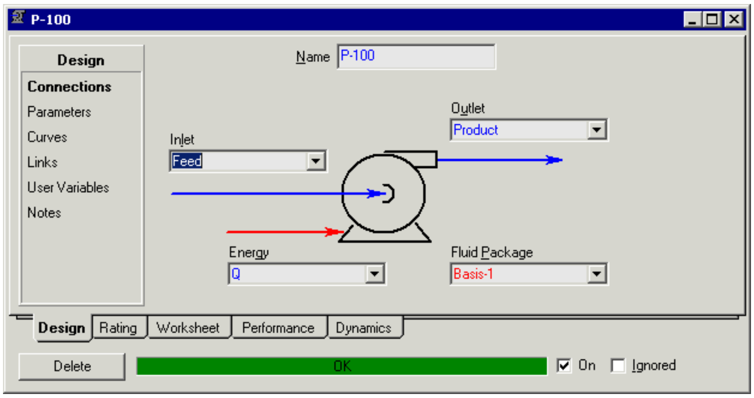
1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.
2. Double-click the **Pump** icon.


Pump icon

The Pump property view appears.

Figure 9.28



The **On** checkbox enables you to toggle between turning on and turning off the pump operation.

- Selected **On** checkbox indicates the Pump is on, and works as normal. (Default setting)
- Cleared **On** checkbox indicates the Pump is off, and the inlet stream passes through the pump operation unchanged. In other words, the outlet stream is exactly the same as the inlet stream.

When you use the **On** option, you should specify a pressure rise rather than specify the pressures of the inlet stream and outlet stream.

If you specify a Delta P, this value is simply ignored when you turn the Pump off.

If you specify the pressures of the inlet stream and outlet stream, you get a consistency error when you turn the Pump off, as HYSYS attempts to pass the inlet stream conditions to the outlet stream.

- In Steady State mode, the **On** checkbox is always available.
- In Dynamics mode, the **On** checkbox is only available for the following situations:
  - Pump speed is being used as a dynamic spec.

This requires that curves are used as a dynamic spec.

- Power is being used as a dynamic spec.
- Electric motor is being used as a dynamic spec.

9.3.3 Design Tab

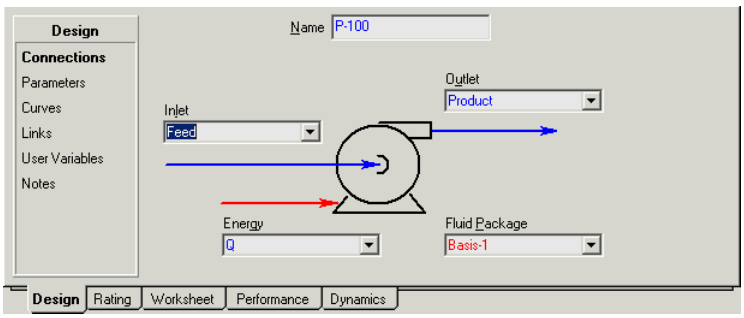
The Design tab consists of the following pages:

- Connections
- Parameters
- Curves
- Links
- User Variables
- Notes

Connections Page

On the Connections page, you can specify the pump name, fluid package, and inlet, outlet, and energy streams of the Pump.

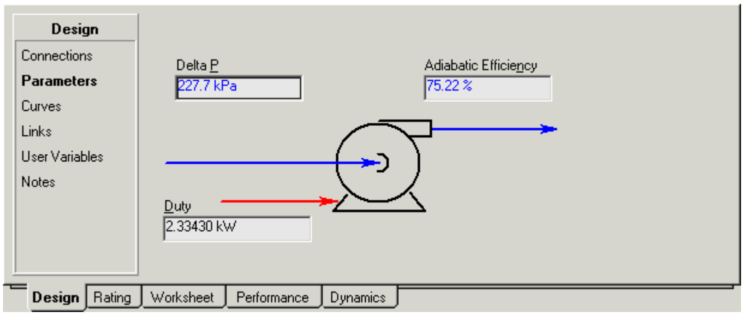
Figure 9.29



Parameters Page

You can specify the adiabatic efficiency, Delta P, and pump energy (power) parameters on the Parameters page.

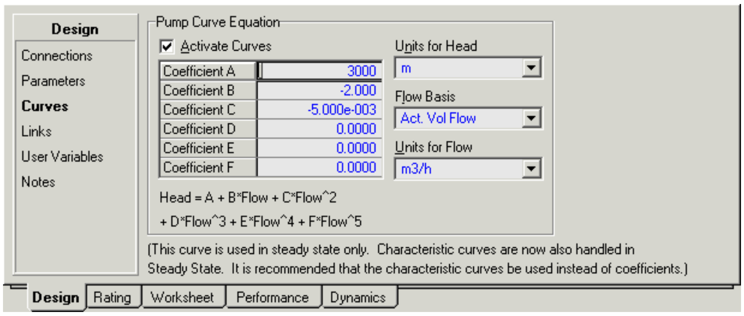
Figure 9.30



Curves Page

The Curves page allows you to configure the pump based on the pump curve. On the Curves page you can create the pump curve using the equation provided.

Figure 9.31



To generate a pump curve:

1. On the **Curves** page, select the units for the Head, Flow Basis, and Flow Rate variables.
2. Enter the coefficients for the quadratic pump equation.

3. Select the **Activate Curves** checkbox.

The Activate Curves checkbox can only be selected if the Use Curves checkbox in the Curves page of the Rating tab is clear.

Based on the calculated pump curve results, HYSYS determines the pressure rise across the Pump for the given flowrate.

To avoid a consistency error, ensure that you have not specified the pressure rise across the Pump, either in the attached streams or in the operation itself.

Phasing out one method

Currently HYSYS Pump operation provides two methods to generate pump curve data in Steady State: curve equation and curve characteristics.

If an old case is loaded into HYSYS and the old case contained converged pumps that use curve equation to generate the pump curves, HYSYS automatically populates a curve characteristic set to generate a new pump curve similar to the old pump curve based on the curve equation specifications.

HYSYS does not automatically replace the new pump curve with the old pump curve.

A warning message also appears informing you about the new pump curve and suggesting that you switch to the curve characteristic method.

HYSYS supports both methods, however the curve equation method will eventually be phased out.

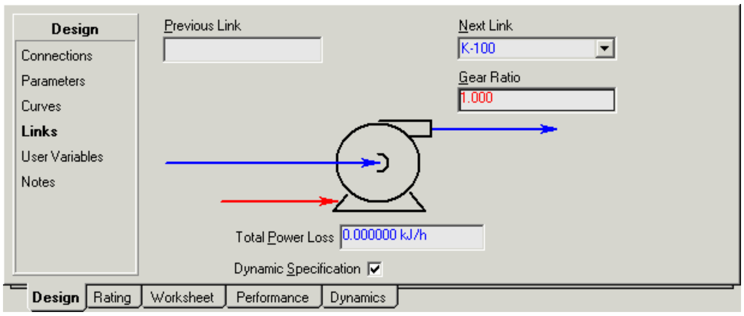
To switch to the curve characteristic method:

1. Open the Pump property view.
2. Click on the **Design** tab and select the **Curves** page.
3. Clear the **Activate Curves** checkbox.
4. Click on the **Rating** tab and select the **Curves** page.
5. Click the **Use Curves** checkbox.
6. Click on the **Design** tab and select the **Parameters** page.
7. Delete any specified values in the **Adiabatic Efficiency** field or **Duty** field.

Links Page

In HYSYS, Pumps can have shafts which are physically connected. The rotational equipment linker operates both in Steady State and Dynamic mode.

Figure 9.32



The following table lists and describes the objects in the Links page:

Object	Description
Previous Link field	Displays the HYSYS rotating equipment operation connected on one side of the shaft.
Next Link drop-down list	Allows you to select a rotating equipment operation to connect on the other side of the shaft.
Gear Ratio field	Displays the ratio of the speed from the next linked operation divided by the speed of the current pump.

Object	Description
Total Power Loss field	Depending on the configuration of the pump and the information specified, you can either: • View the total power loss of the linked operation. • Specify the total power loss of the linked operation.
Dynamic Specification checkbox	Allows you to specify the total power loss (or power gain by entering a negative value) for the linked operation. You can ignore this option in Steady State mode.

It is not significant which order the Pumps are linked. The notion of previous and next links is arbitrary and determined by the user.

Linked Pump operations require curves. In Dynamics mode, to fully define a set of linked operations, you must select the **Use the Characteristic Curves** checkbox for each of the linked Pumps in the **Specs** page of the **Dynamics** tab.

In Dynamics mode when you link rotating operation, the pressure flow equations are affected as follow:

- An energy conservation equation is set such that the sum of the operation powers equals the total power.
- Each pair of operation has their speeds set to equal.

One additional dynamic specification is usually required for the set. The total power loss from the linked operations can be specified. For a series of linked Pumps, it is desired to input a total power:

$$\textit{Total Power Input} = -\textit{Total Power Loss} \tag{9.35}$$

An electric motor connected to the current operation or a linked operation can also supply the total power.

It is possible to link a Pump to a Compressor and use the Pump as a turbine to generate kinetic energy to drive the Compressor. If this option is selected, the total power loss is typically specified as zero.

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

9.3.4 Rating Tab

If you are working exclusively in Steady State mode, you are not required to change any information on most of the pages accessible through the Rating tab. The Rating tab consists of the following pages:

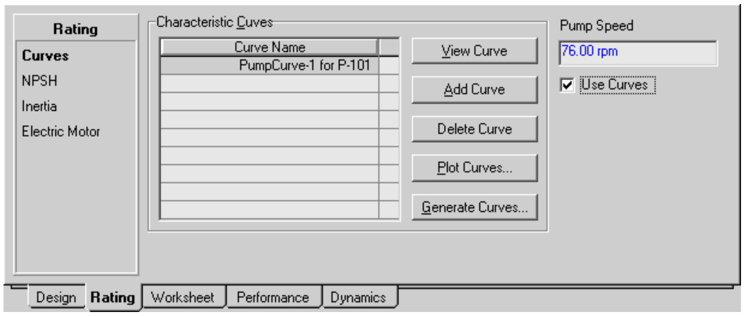
- Curves
- NPSH
- Nozzles
- Inertia
- Electric Motor
- Startup

Curves Page

The Curves page allows you to configure the pump based on a pump curve. The pump curve is generated by curve characteristics.

The curves used on this page work both in steady state and dynamic mode.

Figure 9.33



The curve characteristics consist of pump efficiency, pump head, pump flow rate, and pump speed variables. The pump can be configured based on multiple pump curves and speeds.

The following table lists and describes the objects in the Curves page:

Object	Description
Curve Name column	Displays the names of the curve data for the pump.
View Curve button	Allows you to open the **Curve Property View** and modify the curve characteristic of the selected curve. This button is only available if there is a curve available in the **Curve Name** column.
Add Curve button	Opens the **Curve Property View** and allows you to create a curve.
Delete Curve button	Allows you to delete the selected curve in the **Curve Name** column. This button is only available if there is a curve available in the **Curve Name** column.
Plot Curves button	Allows you to generate a plot of all the curve in the **Curves Profiles Property View**.
Generate Curves button	Allows you to manipulate and generate the curve using the **Generate Curve Options Property View**.
Pump Speed field	Enables you to specify a pump speed for all the curve data.
Use Curves checkbox	Enables you to accept the pump curve data generated by the curve characteristics. The **Use Curves** checkbox can only be selected if the **Activate Curves** checkbox is clear in the **Curves** page of the **Design** tab.

Depending on the type of pump curve you want to generate the following information needs to be supplied for the pump to solve:

- For a performance curve (one curve), the feed temperature and pressure must be supplied along with one of flow, duty, outlet pressure or efficiency.
- For normalized curves, the feed temperature and pressure must be specified along with two of flow, speed, duty, outlet pressure, and efficiency.

For the two variables, either flow and/or speed must be specified. A pump with duty and efficiency specified can not be solved using the curve characteristic option.

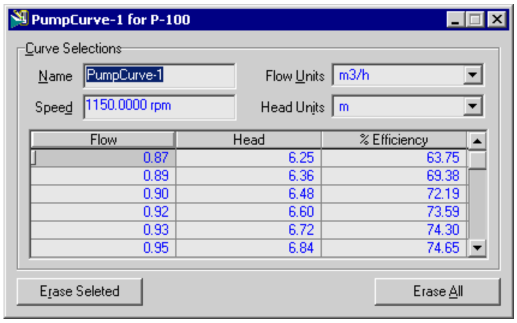
In addition, if outlet pressure or efficiency is supplied as one of the variables (for performance or normalized curves) and their corresponding curve is a parabola or has multiple flows for a given pressure or efficiency, then there may be multiple solutions. HYSYS will notify you of this possibility but will still solve to the first solution only. If iterations are required, basically any problems that do not have both flow and speed specified for a normalized problem or no flow for a performance problem, then HYSYS deploys the Secant method to converge to a solution. The number of maximum iterations is set at 10000 and is not modifiable.

To specify data for a pump curve:

1. On the **Curves** page, click the **Add Curve** button, the Curve property view appears.
2. On the Curve property view, specify the pump speed in the **Speed** field.

3. Specify the flow, head, and %efficiency data points for a single curve in the appropriate cells.

Figure 9.34



4. For each additional curve, repeat step #1 and #2.
   - Click the **Erase Selected** button to delete the entire row (Flow, Head or Efficiency) of the selected cell.
   - Click the **Erase All** button to delete all Flow, Head, and Efficiency data for the curve.
5. After entering all your curve data, click the **Close** icon to return to the Pump property view.

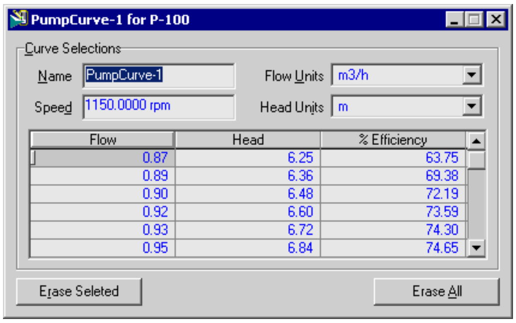
HYSYS uses the curve(s) to determine the appropriate efficiency for your operational conditions. If you specify curves, ensure the Efficiency values on the **Parameters** page are empty or a consistency error is generated.

Curve Property View

You can access the Curve property view by:

- Clicking the **Add Curve** button.
- Selecting a curve data and clicking the **View Curve** button.

Figure 9.35



In the Curve property view, you can specify the following data:

- **Name** field. Name of the Curve.
- **Speed** field. The rotation speed of the Pump. This is optional if you specify only one curve.
- **Flow Units** field. Units for the Flow in Volume/Time units.
- **Head Units** field. Units for the Head in Length units.
- **Flow / Head / Efficiency** cells. Enter any number of data points for the Curve.

The **Erase Selected** button allows you to delete the entire row (Flow, Head or Efficiency) of the selected cell.

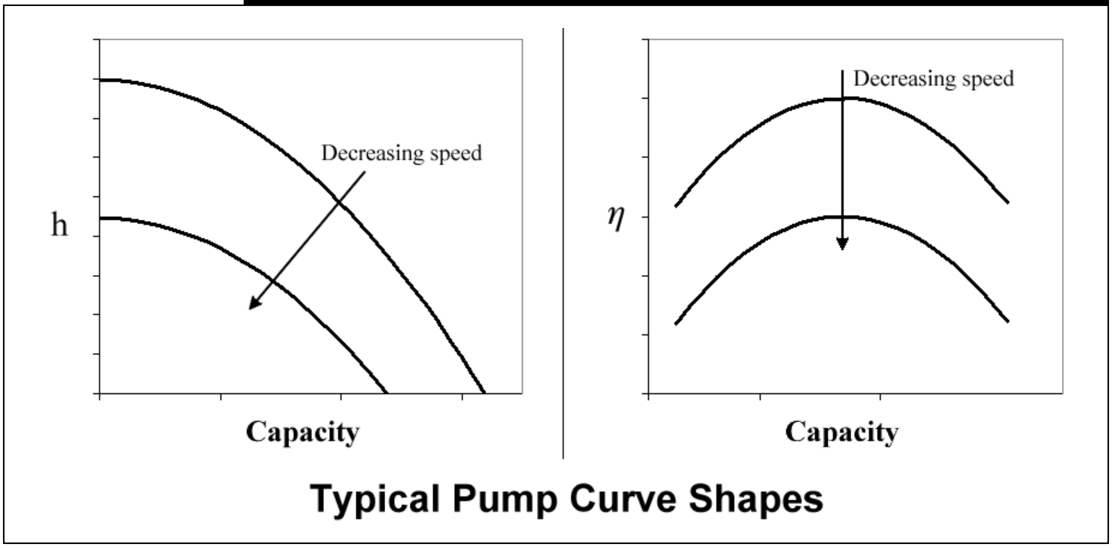
The **Erase All** button allows you to delete all Flow, Head, and Efficiency data for the curve.

HYSYS can interpolate values for the efficiency and head of the Compressor or Expander for speeds that are not plotted.

In order to run a stable and realistic dynamic model, HYSYS requires you to input reasonable curves. If Compressors or Expanders are linked, it is a good idea to ensure that the curves plotted for each unit operation span a common speed and capacity range.

Typical curves are plotted in the figure below.

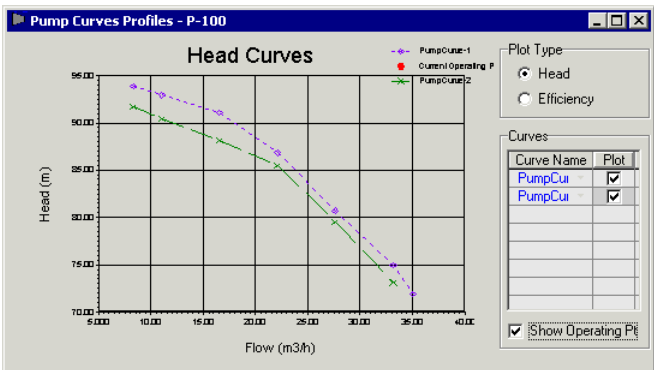
Figure 9.36



Curves Profiles Property View

The Curves Profiles property view allows you to see the plot of the curve data.

Figure 9.37



To access the Curves Profiles property view, click the **Plot Curves** button on the **Curves** page in the **Rating** tab of the Pump property view.

The following table lists and describes the objects available in the Curves Profiles property view:

Object	Description
Plot	Displays the selected curve data in plot format.
Head radio button	Allows you to view the Head vs. Flow curve data plot.
Efficiency radio button	Allows you to view the Efficiency vs. Flow curve data plot.
Curve Name column	Displays the names of the curve data available for the plot.
Plot checkboxes	Allows you to toggle between displaying and hiding the associate curve data in the plot.
Show Operating Pt checkbox	Allows you to toggle between displaying and hiding the curve data generated by the Pump's current operating conditions and specifications.

Generate Curve Options Property View

The Generate Curve Options property view allows you to generate curve data based on the specified pump design parameters. HYSYS automatically generates three curves based on three different speeds: user specified speed, user specified speed multiplied by low speed %, and user specified speed multiplied by low low speed %.

Each curve is generated using the following data point assumptions:

- A point based on the Head of the pump, Capacity of the pump, and the assumption that shutoffhead (in other words, at 0 flow) occurs at 110% of the design Head (110% is the design Head factor).
- A point based on the Head of the pump, Capacity of the pump, and the assumption that maximum flow (in other words, at 0 Head) occurs at 200% of the design capacity (200% is the design flow factor).
- A point based on the following expression:

$$(\text{capacity,efficiency}) = (0,\text{design efficiency factor} \times \text{efficiency design}) \tag{9.36}$$

- A point based on the following expression:

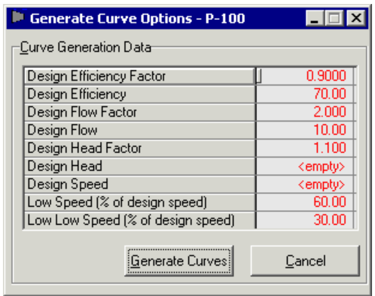
$$(\text{capacity}, \text{efficiency}) = (\text{design flow}, \text{efficiency design}) \tag{9.37}$$

- A point based on the following expression:

$$\begin{aligned}(\text{capacity}, \text{efficiency}) &= (\text{design flow factor} \times \text{design flow},\\ &\text{efficiency design} \times \text{design efficiency factor})\end{aligned} \tag{9.38}$$

To access the Generate Curve Options property view, click the **Generate Curves** button on the **Curves** page in the **Rating** tab of the Pump property view.

Figure 9.38



The following table lists and describes the objects available in the Generate Curve Options property view:

Object	Description
Design Efficiency Factor cell	Allows you to manipulate the pump design efficiency factor. Default value is 0.90.
Design Efficiency cell	Allows you to manipulate the design efficiency of the pump. HYSYS provides a default value of 70%.
Design Flow Factor cell	Allows you to manipulate the pump design flow factor. Default value is 2.
Design Flow cell	Allows you to manipulate the pump design flow. Default value is 10.
Design Head Factor cell	Allows you to manipulate the pump design Head factor. Default value is 1.10.
Design Head cell	Allows you to specify the design Head of the pump.
Design Speed cell	Allows you to specify the design speed of the pump.

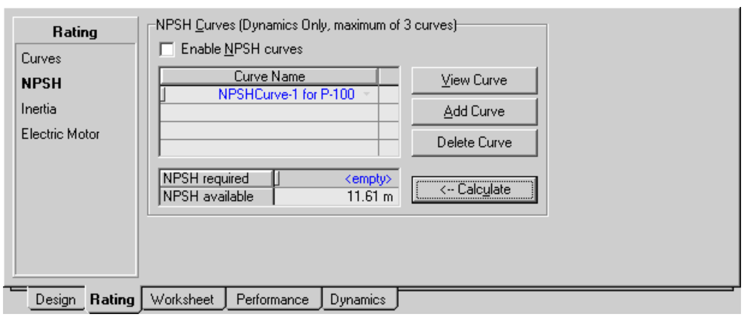
Object	Description
Low Speed cell	Allows you to manipulate the pump low speed based on the percentage value of the pump design speed. Default value is 60%.
Low Low Speed cell	Allows you to manipulate the pump low low speed based on the percentage value of the pump design speed. Default value is 30%.
Generate Curves button	Allows you to generate the curve data based on the specified pump design. Any previous specified curve data in the **Curves** page of the **Rating** tab will be deleted, when you generate the new curve data.
Cancel button	Allows you to exit the Generate Curves Options property view without generating any curve data.

NPSH Page

Net Positive Suction Head (NPSH) is an important factor to consider when choosing a Pump. Sufficient NPSH is required at the inlet of the Pump to prevent the formation of small bubbles in the pump casing which can damage the Pump. This is known as cavitation. For a given Pump, the net positive suction head required to prevent cavitation, $NPSH_{required}$, is a function of the capacity (volumetric flowrate) and speed of the Pump.

In HYSYS, NPSH curves can be specified like regular pump curves on the NPSH page.

Figure 9.39



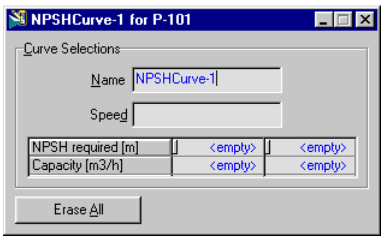
To add or edit a NPSH curve from the NPSH page:

1. Select the **Enable NPSH curves** checkbox.
2. Click the **Add Curve** button, the NPSH Curve property view appears.
3. Specify the speed for each curve.
4. Enter a capacity and NPSH for two points on the curve. Only two points are required for the NPSH curves since:

$$\log(NPSH_{required}) \propto \log(capacity) \tag{9.39}$$

5. To remove all the data points, click the **Erase All** button.

Figure 9.40



6. For each additional curve, repeat steps #2 to #4.

The $NPSH_{required}$ value can either be taken from the NPSH curves or specified directly in the NPSH required field. To directly specify the $NPSH_{required}$, you must first clear the **Enable NPSH curves** checkbox.

$NPSH_{available}$ can be explicitly calculated from the flowsheet conditions by clicking the Calculate Head button. The $NPSH_{available}$ is calculated as follows:

$$NPSH_{available} = \frac{P_1 - P_{vap}}{\rho g} + \left(\frac{V_1^2}{2g}\right) \tag{9.40}$$

where:

P_1 = *inlet stream pressure to the pump*

P_{vap} = *vapour pressure of the inlet stream*

ρ = *density of the fluid*

V_1 = *velocity of the inlet stream*

g = *gravity constant*

To prevent pump cavitation the $NPSH_{available}$ must be above the $NPSH_{required}$. If a pump cavitates in HYSYS, it is modeled by scaling the density of the fluid, ρ, randomly between zero and one.

Nozzles Page

Refer to **Section 1.3.6 - Nozzles Page** for more information.

The Nozzles page contains information regarding the elevation and diameter of the nozzles.

For a Pump unit operation it is strongly recommended that the elevation of the inlet and exit nozzles are equal. If you want to model static head, the entire piece of equipment can be moved by modifying the Base Elevation relative to Ground Elevation field.

Inertia Page

Refer to **Section 1.6.4 - Inertia** in the **HYSYS Dynamic Modeling** guide for more information.

The inertia modeling parameters and the frictional loss associated with the impeller in the Pump can be specified on this page. The HYSYS Dynamics license is required to use the Inertia features.

Electric Motor Page

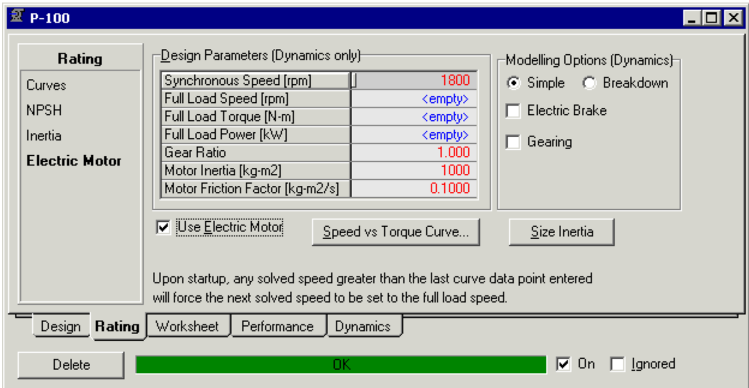
The Electric Motor page allows you to drive your rotating unit operation through the designation of a motor torque versus speed curve. These torque vs. speed curves can either be obtained from the manufacturer for the electric motor being used or from a typical curve for the motor type. For most process industry applications, a NEMA type A or B electric motor is used. When you use the Electric Motor option the torque (and power) generated by the motor is balanced against the torque consumed by the rotating equipment.

The Electric Motor functionality is only relevant in Dynamics mode.

The Electric Motor option uses one degree of freedom in your dynamic specifications.

The results of the Electric Motor option are presented on the Power Page in the Performance Tab of the rotating equipment operation.

Figure 9.41



The following table lists and describes the objects in the Electric Motor page:

Object	Description
Synchronous Speed cell	Allows you to specify the synchronous speed of the motor.
Full Load Speed cell	Allows you to specify the design speed of the motor.
Full Load Torque cell	Allows you to specify the design torque of the motor.
Full Load Power cell	Allows you to specify the design power of the motor.
Gear Ratio cell	Allows you to manipulate the gear ratio. The gear ratio is the rotating equipment's speed divided by the motor speed.
Motor Inertia cell	Allows you to specify the motor inertia.
Motor Friction Factor cell	Allows you to specify the motor friction factor.
User Electric Motor checkbox	Allows you to toggle between using or ignoring the electric motor functionality.

Refer to **Operation Model** section for more information.

Object	Description
Speed vs Torque Curve button	Allows you to view the plot and specify the data in the **Speed vs. Torque Curve Property View**.
Size Inertia button	Allows you to calculate the inertia based on the following equation: $$I = 0.0043\left(\frac{P}{N}\right)^{1.48}$$ *where:* I = *inertia* ($kg \cdot m^3$) P = *full load power of the motor (kW)* N = *full load speed of the motor (rpm/1000)*
Simple radio button	Allows you to select the **Simple** model for the modelling option.
Breakdown radio button	Allows you to select the **Breakdown** model for the modelling option.
Electric Brake checkbox	Allows you to model the torque force on the rotating equipment simply by changing the sign of the produced torque value.
Gearing checkbox	Allows the gear ratio to be updated during integration. A zero value for the gear ratio indicates a decoupling of the equipment.

Theory

The definition of torque is found from the following equation:

$$T = \frac{P \times \omega \times 2 \times \pi}{1000 \times 60} \tag{9.41}$$

where:

P = *power consumption (kW)*

T = *torque (Nm)*

ω = *synchronous speed (rpm)*

The synchronous electric motor speed can be found from:

$$\omega = \frac{120f}{p} \qquad (9.42)$$

where:

f = power supply frequency (Hz), typically either of 50 or 60

p = number of poles on the stator

The number of poles is always an even number of 2, 4, 6, 8, 10, and so forth. In North America, common motor speeds are always 3600, 1800, 1200, 900, 720, and so forth.

The relationships of inertia and friction loss in the total energy balance are the same as for the pump and compressor operations.

Operation Model

There are three ways to use the Electric Motor curve, each with progressing rigor.

- **Simple Model**. The easiest calculation is the Simple modelling option (default). This model is useful if you just want to model the startup/shutdown transient and want to keep the equipment at the fixed full load speed once operating. In this mode, once the speed has accelerated enough to become larger than the last (largest) curve speed value entered, the motor speed immediately is set to the full load speed and remains there until the motor is turned off. If the process invokes a larger torque than the motor curve suggests the motor can produce, the speed still remains synchronous and remains at its full load value.
- **Breakdown Model**. The Breakdown modelling option allows the speed to reduce if the system torque or resistance gets too large. To use this option, the largest (last) curve speed value entered should be just less than the full load speed value. This provides for a smooth transition in operation. You can drop the curve to a lower torque than the breakdown torque if desired.
- **Simple Model Modified**. The third modelling approach is to use the Simple model option, but enter the speed vs torque curve up to a speed value of 99.99% of the synchronous speed. In this case the full load speed

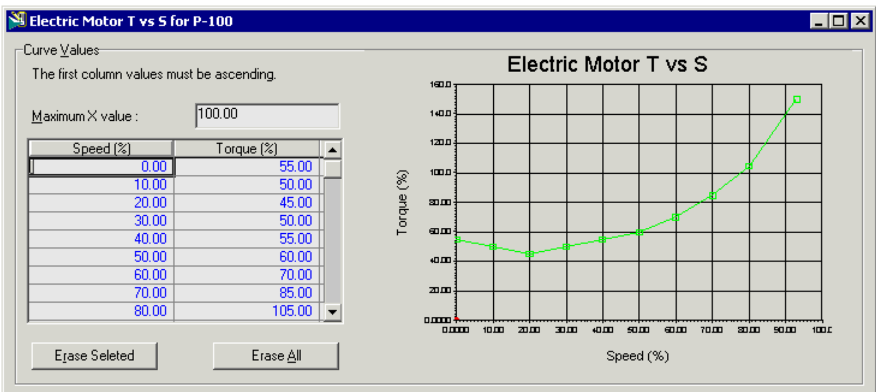
entered is only used, if necessary, to calculate the full load torque and is not used otherwise. With this approach, the speed vs torque curve must ascend or drop from the breakdown torque to approach zero torque at 100% speed. For most motor types, this approach is nearly vertical (asymptotic). This modelling approach allows for the simulation of the slippage of the motor speed based upon the actual and current system resistance. The operating speed of the motor will then move based upon the process model operation. Use a near vertical curve to keep a constant speed or level it off more to allow greater slip. This performance should be predicted by using an accurate manufacturers torque vs speed curve.

The speed and torque are not solved simultaneously with the pressure flow solution but instead is lagged by a time step. You may need to use a smaller time step to ensure accuracy and pressure flow solver convergence.

Speed vs. Torque Curve Property View

The Speed vs. Torque Curve property view displays the data curve of speed versus torque in both table and plot format. To access the Speed vs. Torque Curve property view, click the **Speed vs Torque Curve** button on the **Electric Motor** page of the **Rating** tab.

Figure 9.42



The values under the Speed and Torque columns are entered as a percent of the Full Load values.

The Speed vs. Torque curve must always contain a 0% speed value.

During integration, the current operating point appears on the Torque vs. Speed curve.

The maximum table speed cannot be greater than the value in the Maximum X value field. The Maximum X value is the ratio of the full load speed to the synchronous speed.

The following table lists and describes the objects available in the Speed vs. Torque Curve property view:

Object	Description
Maximum X value display field	Displays the ratio of the full load speed to the synchronous speed.
Speed column	Allows you to specify speed percentage values you want to plot.
Torque column	Allows you to specify the torque percentage values associated with the speed.
Erase Selected button	Allows you to delete the row containing both speed and torque percentage values of the selected cell.
Erase All button	Allows you to delete all the values in the table.

Design Page

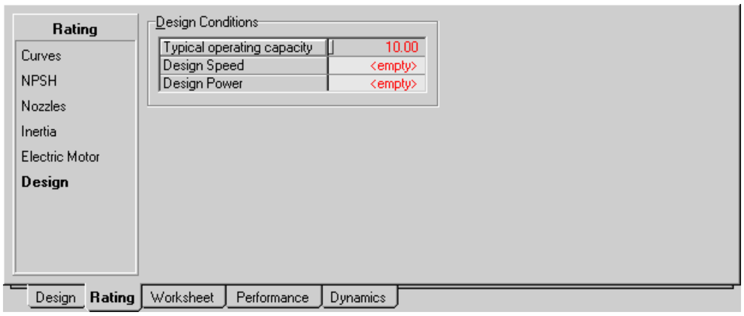
The Design page allows you to specify the typical or design operating capacity, pump speed, and power consumption.

- **Typical operating capacity** cell allows you to specify a value used to assist pump start up priming when vapor is present in the inlet stream.
- **Design Speed** cell allows you to specify the speed value used for the Auto Pump Curve generation feature and the pump inertia sizing.
- **Design Power** cell allows you to specify the power value used for the Auto Pump Curve generation feature and for pump inertia sizing.

Refer to **Section 1.6.6 - Design** in the **HYSYS Dynamic Modeling** guide for more information.

The HYSYS Dynamics license is required to use the options in the Design page.

Figure 9.43



9.3.5 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

The PF Specs page is relevant to dynamics cases only.

9.3.6 Performance Tab

The Performance tab contains the calculated results of the pump.

Results Page

The Results page contains pump head information. The values for total head, pressure head, velocity head, Delta P excluding static head, total power, friction loss, rotational inertia, and fluid power are calculated values.

The Total Head field is used only for dynamic simulation.

Power Page

The Power page displays the following calculated values:

- pump rotor power variables
- pump rotor torque variables
- electric motor power variables (if applicable)
- electric motor torque variables (if applicable)
- other electric motor data (if applicable)

9.3.7 Dynamics Tab

The Dynamics tab is used only for dynamic simulation. The Dynamic tab contains the following pages:

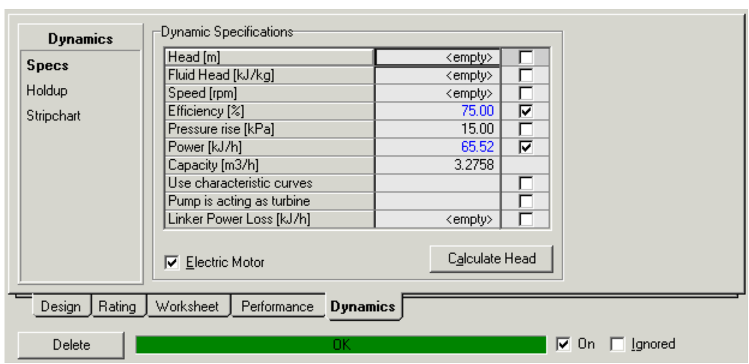
- Specs
- Holdup
- Stripchart

If you are working exclusively in Steady State mode, you are not required to change any of the values on the pages accessible through the Dynamics tab.

Specs Page

The dynamic specifications of the Pump can be specified on the Specs page.

Figure 9.44



In general, two specifications should be selected in the Dynamics Specifications group in order for the Pump operation to fully solve. You should be aware of specifications, which may cause complications or singularity in the pressure flow matrix. Some examples of such cases are:

- The **Pressure rise** checkbox should not be selected if the inlet and exit stream pressures are specified.
- The **Speed** checkbox should not be selected if the **Use Characteristic Curves** checkbox is not selected.

The possible dynamic specifications are as follows:

Head

The ideal head, h, can easily be defined as a function of the isentropic or polytropic work. The relationship is:

$$W = (MW)Fgh \tag{9.43}$$

where:

W = ideal pump power

MW = molecular weight of the gas

F = molar flow rate of the inlet stream

g = gravity acceleration

or using **Equation (9.34)**, the head is defined as:

$$h = \frac{P_2 - P_1}{\rho g} \tag{9.44}$$

If pump curves are provided in the Curves page of the Rating tab, the ideal head can be interpolated from the flow of gas and the speed of the pump.

Fluid Head

The Fluid Head is the produced head in units of energy per unit mass.

Speed

The rotational speed of the shaft, ω, driving the Pump can be specified.

Efficiency

The efficiency is given as the ratio of the ideal power required by the pump to the actual energy imparted to the fluid. The efficiency, η, is defined as:

$$\eta = \frac{W}{F(MW)(h_2 - h_1)} \tag{9.45}$$

The ideal power required by the pump is provided in **Equation (9.34)**.

The general definition of the efficiency does not include the losses due to the rotational acceleration of the shaft and seal losses. Therefore, the efficiency equations in dynamics are not different at all from the general efficiency equations defined in **Section 9.3.1 - Theory**.

If pump curves are provided in the Curves page of the Rating tab, the efficiency can be interpolated from the flow of gas and the speed of the Pump.

Pressure Rise

A Pressure Rise specification can be selected, if the pressure drop across the Pump is constant.

Power

The duty is defined as the power required to rotate the shaft and provide energy to the fluid. The duty has three components:

$$\text{Duty} = \text{Power supplied to the fluid} + \text{Power required to change the rotational speed of the shaft} + \text{Power lost due to mechanical friction loss} \tag{9.46}$$

The duty should be specified only if there is a fixed rate of energy available to be used to drive the shaft.

Capacity

The capacity is defined as the actual volumetric flow rate entering the Pump.

Use Characteristic Curves

Select the **Use Characteristic Curves** checkbox, if you want to use the curve(s) specified in the Curves page of the Rating tab. If a single curve is specified in a dynamics Pump, the speed of the Pump is not automatically set to the speed of the curve. A different speed can be specified, and HYSYS extrapolates values for head and efficiency.

Pump is acting as turbine

Select the **Pump is acting as turbine** checkbox, if you want the pump to act as a turbine with a pressure drop from inlet to outlet.

Linker Power Loss

Select the **Linker Power Loss** checkbox, if you want to specify the power loss (negative for a power gain) of the linked operations.

Electric Motor

Select the **Electric Motor** checkbox if you want to use the electric motor functionality.

Holdup Page

Refer to **Section 1.3.3 - Holdup Page** for more information.

Typical pumps in actual plants usually have significantly less holdup than most other unit operations in a plant. Therefore, the volume of the Pump operation in HYSYS cannot be specified, and is assumed to be zero on the Holdup page.

Stripchart Page

Refer to **Section 1.3.7 - Stripchart Page/Tab** for more information.

The Stripchart page allows you to select and create default strip charts containing various variable associated to the operation.

9.4 References

[1] Gas Processors Association. Gas Processors Suppliers Association (1998) p.13-1 to p13-20

[2] Campbell M.John. Gas Conditioning and Processing (vol2) 7th edi. 1994, p213-221

10 Separation Operations

10.1 Component Splitter

With a Component Splitter, a material feed stream is separated into two component streams based on the parameters and split fractions that you specify. You must specify the fraction of each feed component that exits the Component Splitter into the overhead product stream. Use it to approximate the separation for proprietary and non-standard separation processes that are not handled elsewhere in HYSYS.

10.1.1 Theory

The Component Splitter satisfies the material balance for each component:

$$f_i = a_i + b_i \tag{10.1}$$

where:

f_i = *molar flow of the ith component in the feed*

a_i = *molar flow of the ith component in the overhead*

b_i = *molar flow in the ith component in the bottoms*

The molar flows going to the overhead and bottoms are calculated as:

$$a_i = x_i f_i \tag{10.2}$$

$$b_i = (1-x_i)\, f_i \tag{10.3}$$

where:

x_i = *split, or fraction of component i going to the overhead*

Once the composition, vapour fraction, and pressure of the outlet streams are know, a P-VF flash is performed to obtain the temperatures and heat flows.

An overall heat balance is performed to obtain the energy stream heat flow:

$$h_E = h_F - h_O - h_B \tag{10.4}$$

where:

h_E = *enthalpy of unknown energy stream*

h_F = *enthalpy of feed stream*

h_O = *enthalpy of overhead stream*

h_B = *enthalpy of bottoms stream*

10.1.2 Component Splitter Property View

There are two ways that you can add a Component Splitter to your simulation:

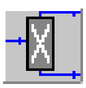
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Short Cut Columns** radio button.
3. From the list of available unit operations, select the **Component Splitter** model.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

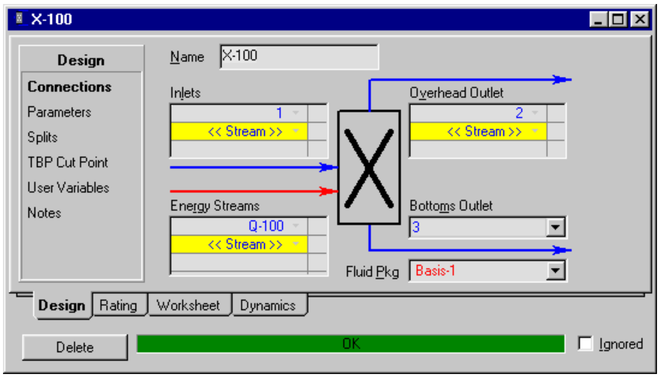
   You can also open the Object Palette by pressing **F4**.
2. Double-click the **Component Splitter** icon.



Component Splitter icon

The Component Splitter property view appears.

Figure 10.1



10.1.3 Design Tab

The Design tab contains the following pages:

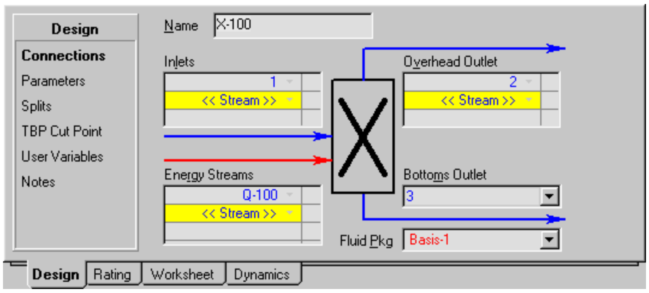
- Connections
- Parameters
- Splits
- TBP Cut Point
- User Variables
- Notes

Each of the pages are discussed in the following sections.

Connections Page

You can specify an unlimited number of inlet streams to the Component Splitter on the Connections page. You must specify the overhead outlet stream, bottoms outlet stream, and an unlimited number of energy streams.

Figure 10.2

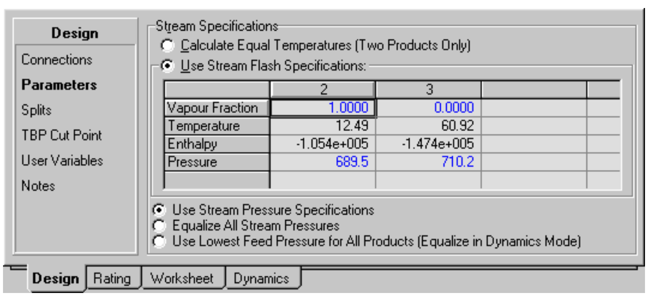


One of the attached energy streams should have an unspecified energy value to allow the operation to solve the energy balance.

Parameters Page

The Parameters page displays the stream parameters. You must specify the stream parameters, which include the vapour fraction, pressure of the overhead stream, and pressure of the bottoms stream.

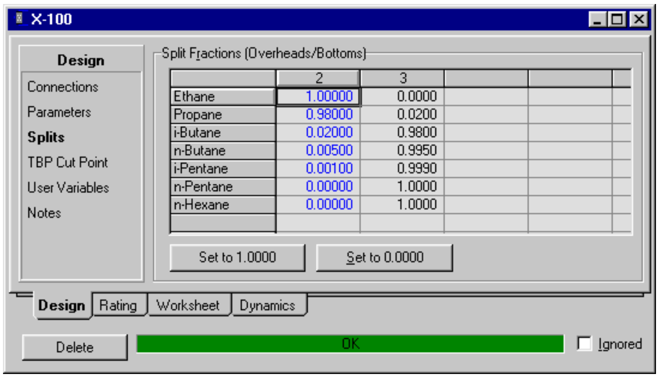
Figure 10.3



Splits Page

The Splits page allows you to specify the separation fraction of the outlet streams.

Figure 10.4



The Splits, or separation fractions ranging from 0 to 1, must be specified for each component in the overhead stream exiting the Component Splitter. The quantity in the bottoms product is set once the overhead fraction is known.

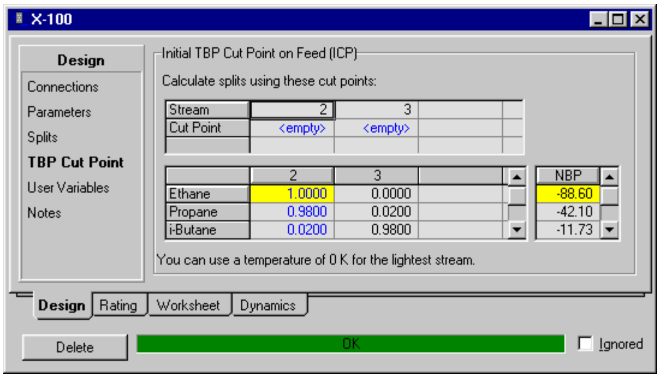
The two buttons on the Splits page, All 1 and All 0, allow you to specify overhead fractions of one (100%) or zero (0%), respectively, for all components. These buttons are useful if many components are leaving entirely in either the overhead stream or bottoms stream.

For example, if the majority of your components are going overhead, simply click the All 1 button, rather than repeatedly entering fractions of 1. Then, correct the splits appropriately for the components not leaving entirely in the overhead.

TBP Cut Point Page

The TBP Cut Point page allows you to specify the compositions of the product streams by providing the TBP Cut Point between the streams, and assuming that there is sharp separation at the cut point.

Figure 10.5



You can specify a temperature cut point of 0 K and higher.

On the TBP Cut Point page, the upper table allows you to specify the initial TBP Cut Point on the Feed for each product stream except for the overhead. The Initial Cut Point values are expressed in temperature and they are listed in ascending order. Consecutive streams can have the same Initial Cut Point value, implying that the second or subsequent stream has zero flow.

The TBP Cut Point Page is designed for streams with hypothetical components.

The bottom table allows you to specify the split fraction for each component in the stream. The split fraction values are also available on the Splits page of the Design tab.

Pure components are distributed according to their NBP (Natural Boiling Point) while the TBP (True Boiling Point) of the pure components defines the boundaries of distribution. The NBP for each component is displayed in the NBP table for reference.

The TBP Cut Point page is designed for handling streams that carry hypocomponents. The hypocomponents are treated as a continuum and they are distributed according to their FBP (Final Boiling Point). The FBP of each hypocomponent is first calculated by sorting the NBP of the hypocomponents in ascending order. Then with the sorted order, the FBP of the last hypocomponent is calculated as follows:

$$FBP_{last} = NBP_{last} + (NBP_{last} - NBP_{last-1}) \tag{10.5}$$

The FBPs for other hypocomponents is calculated by:

$$FBP_i = \frac{NBP_i + NBP_{i+1}}{2} \tag{10.6}$$

The hypocomponents are then distributed according to where the cut point lies. The boiling range for each hypocomponent is defined by the FBP of the previous component to the FBP of the current component. The boiling range for each product stream is from its Initial TBP Cut Point to the Initial TBP Cut Point of the next stream.

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

10.1.4 Rating Tab

You cannot provide any information for the Component Splitter on the Rating tab when in steady state.

Nozzles Page

For more information, refer to **Section 1.3.6 - Nozzles Page**.

The Nozzles page contains information regarding the elevation and diameter of the nozzles.

10.1.5 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

The PF Specs page is relevant to dynamics cases only.

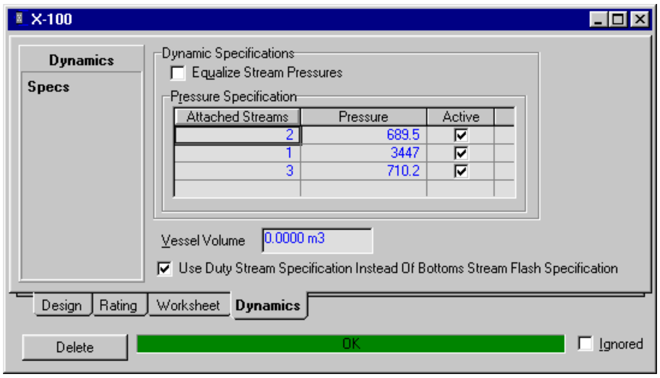
10.1.6 Dynamics Tab

Information available on this page is relevant only to cases in Dynamic mode. The Dynamics tab consists of the Specs page.

Specs Page

The Specs page contains information regarding pressure specifications of the streams.

Figure 10.6



The Equal Pressures checkbox allows you to propagate the pressure from one stream to all others. If you want to equalize the pressures you have to free up the pressure specs on the streams that you want the pressure to be propagated to.

The Vessel Volume is also specified on this page. This volume is only used to affect the compositional affects in this operation. The Pressure - Flow and hydraulics are not affected by this value. This volume is only a simplified and synthetic way of getting some lag in the compositional affects. The thermal state (temperature and enthalpy) of the outlet streams may or may not be affected by this value depending on the other specifications of this operation. For example, if you have specified the outlet stream temperatures directly, then this volume has no affect, on the other hand if the operation is doing some flash with an external duty (of value zero or otherwise), then this volume has an affect.

10.2 Separator, 3-Phase Separator, & Tank

The property views for the Separator, 3-Phase Separator, and Tank are similar, therefore, the three unit operations are discussed together in this section.

All information in this section applies to the Separator, the 3-Phase Separator, and the Tank operation, unless indicated otherwise.

There is an Operation Type toggle option on the Parameters page of the Design tab that allows you to easily switch from one of these operations to another. For example, you may want to change a fully defined Separator to a 3-Phase Separator. Simply, select the appropriate radio button in the Operation Type toggle option. The only additional information required would be to identify the additional liquid stream. All of the original characteristics of the operation (Parameters, Reactions, and so forth) are retained.

The key differences in the three separator operations are the stream connections (related to the feed separation), which are described in the table below.

Unit Operation	Description
Separator	Multiple feeds, one vapour and one liquid product stream. In Steady State mode, the Separator divides the vessel contents into its constituent vapour and liquid phases.
3-Phase Separator	Multiple feeds, one vapour and two liquid product streams. The 3-Phase Separator operation divides the vessel contents into its constituent vapour, light liquid, and heavy liquid phases.
Tank	Multiple feeds, one liquid and one vapour product stream. The Tank is generally used to simulate liquid surge vessels.

In Dynamic mode, the following unit operations all use the holdup model and therefore, have many of the same properties. Vessel operations in HYSYS have the ability to store a significant amount of holdup.

The key differences in the vessel operations are outlined in the table below.

Unit Operation	Description
Separator	The Separator can have multiple feeds. There are two product nozzles: • liquid • vapour.
3-Phase Separator	The 3-Phase Separator can have multiple feeds. There are 3 product nozzles: • light liquid • heavy liquid • vapour.
Tank	The Tank can have multiple feeds. There are two product nozzles which normally removes liquid and vapour from the Tank.
Condenser	The condenser has one vapour inlet stream. The number and phase of each exit stream depends on the type of condenser. The condenser has a unique method of calculating the duty applied to its holdup.
Reboiler	The reboiler has one liquid inlet stream. The reboiler can have a number of liquid and vapour exit streams.
Reactors	Reactor operations can have multiple inlet and exit streams.
Heat Exchanger (Simple Rating Model, Detailed)	A shell or tube with a single pass in the heat exchanger unit operation can be modeled with a liquid level. Both the shell and tube sides of the heat exchanger have one inlet and one exit stream.

Every dynamic vessel operation in HYSYS has some common features including:

- The geometry of the vessel and the placement and diameter of the attached feed and product nozzles have physical meaning.
- A heat loss model which accounts for the convective and conductive heat transfer that occurs across the vessel wall.
- Various initialization modes which allow you to initialize the vessel at user-specified holdup conditions before running the integrator.
- Various Heater types which determine the way in which heat is transferred to the vessel operation.

10.2.1 Theory

A P-H flash is performed to determine the product conditions and phases. The pressure at which the flash is performed is the lowest feed pressure minus the pressure drop across the vessel. The enthalpy is the combined feed enthalpy plus or minus the duty (for heating, the duty is added; for cooling, the duty is subtracted).

As well as standard forward applications, the Separator and 3-Phase Separator have the ability to back-calculate results. In addition to the standard application (completely defined feed stream(s) being separated at the vessel pressure and enthalpy), the Separator can also use a known product composition to determine the composition(s) of the other product stream(s), and by a balance the feed composition.

In order to back-calculate with the Separator, the following information must be specified:

- One product composition.
- The temperature or pressure of a product stream.
- Two (2-phase Separators) or three (3-phase Separators) flows.

If you are using multiple feed streams, only one feed stream can have an unknown composition in order for HYSYS to back-calculate.

Energy Balance

In Steady State mode Separator energy balance is defined below:

$$H_{feed} \pm Duty = H_{vapour} + H_{heavy} + H_{light} \tag{10.7}$$

where:

H_{feed} = *heat flow of the feed stream(s)*

H_{vapour} = *heat flow of the vapour product stream*

H_{light} = *heat flow of the light liquid product stream*

H_{heavy} = *heat flow of the heavy liquid product stream*

Physical Parameters

The Physical Parameters associated with this operation are the pressure drop across the vessel and the vessel volume.

The default pressure drop across the vessel is zero.

The pressure drop across the vessel is defined as:

$$P = P_{\mathrm{l}} = P_{\mathrm{feed}} - \Delta P = P_{\mathrm{head}} + P_{\mathrm{v}} \tag{10.8}$$

where:

P = *vessel pressure*

P_v = *pressure of vapour product stream*

P_l = *pressure of liquid product stream(s)*

P_{feed} = *pressure of feed stream*
the pressure is assumed to be the lowest pressure of all the feed streams

ΔP = *pressure drop in vessel*

P_{head} = *pressure of the static head*

The vessel volume, together with the set point for liquid level/flow, defines the amount of holdup in the vessel. The amount of liquid volume, or holdup, in the vessel at any time is given by the following expression

$$Holdup = Vessel\ Volume \times \frac{PV(\%Full)}{100} \qquad (10.9)$$

where:

$PV(\%Full)$ = *liquid level in the vessel at time t*

The Vessel Volume is necessary in steady state when modeling a Reactor (CSTR), as it determines the residence time.

Ideal vs. Real

In ideal separators, complete/perfect separation between the gas and liquid phases is assumed.

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 (for example, mesh pads, vane packs, weirs) to reduce the carry over of entrained liquids or gases.

By default the HYSYS separators are ideal separators, however, you can modify the separators to model imperfect separation by using the HYSYS Real Separator capabilities. The real separator offers you a number of advantages:

- Carry Over options so that your model matches your process mass balance or separator design specifications.
- Options to predict the effect of feed phase dispersion, feed conditions, vessel geometry, and inlet / exit devices on carry over.

Refer to **Section 10.2.5 - Rating Tab** for information on the options used to configure a real separator.

10.2.2 Separator General Property View

There are two ways that you can add a Separator, 3 -Phase Separator, or Tank to your simulation:

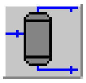
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

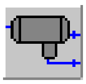
   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Vessels** radio button.
3. From the list of available unit operations, select the **Separator**, **3 Phase Separator**, or **Tank**.
4. Click the **Add** button.

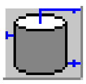
OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.
2. Double-click the **Separator** icon or **3 Phase Separator** icon or **Tank** icon.

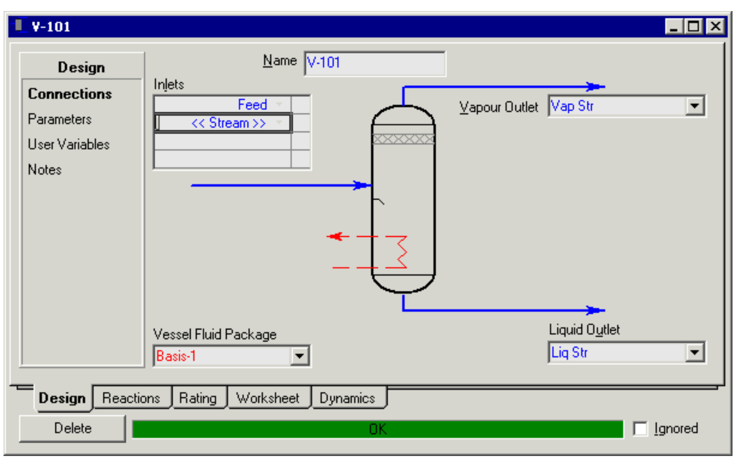

Separator icon


3-Phase Separator icon


Tank icon

The Separator or 3 Phase Separator or Tank property view appears.

Figure 10.7



If you want to use the Separator as a reactor, you can either install a Separator or choose General Reactor from the UnitOps property view.

10.2.3 Design Tab

The Design tab contains options for configuring the separator operation.

Connections Page

The Connections page allows you to specify the streams flowing into and out of the separator operation, the name of the separator operation, and the fluid package associated to the separator operation.

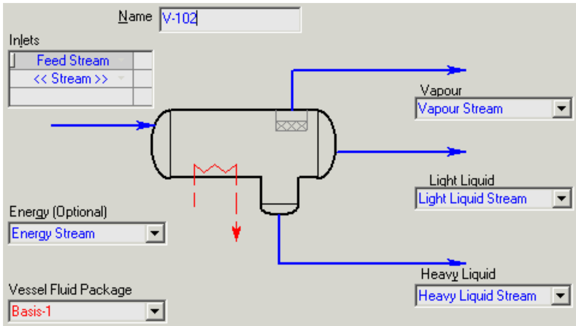
Any of the HYSYS separator operations accept multiple feed streams, as well as an optional energy stream.

Depending on the type of heat transfer you want to make available for the separator operation, the options available in the **Connections** page varies. You are required, however, to always specify the following variables:

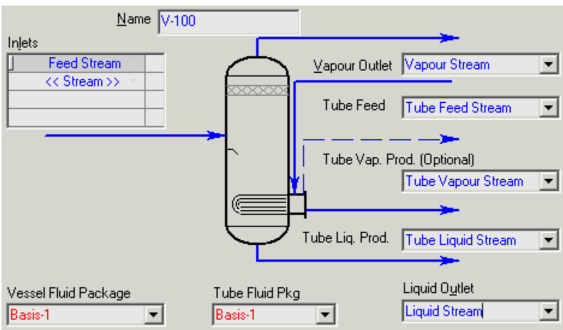
- Name of the separator operation
- Name of the feed streams
- Name of the vapor product stream
- Name of the liquid product stream(s)
- Name of the fluid package of the separator operation

The figure below shows the Connections pages for the three separator operations:

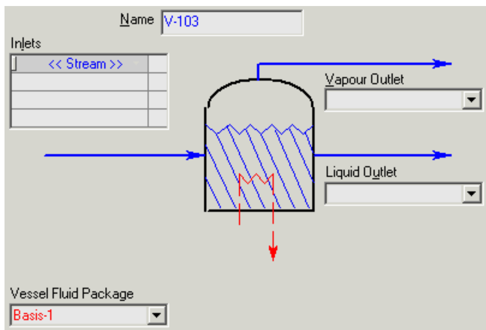
Figure 10.8



Three Phase Separator with energy stream attached.



Separator with kettle reboiler or chiller attached.

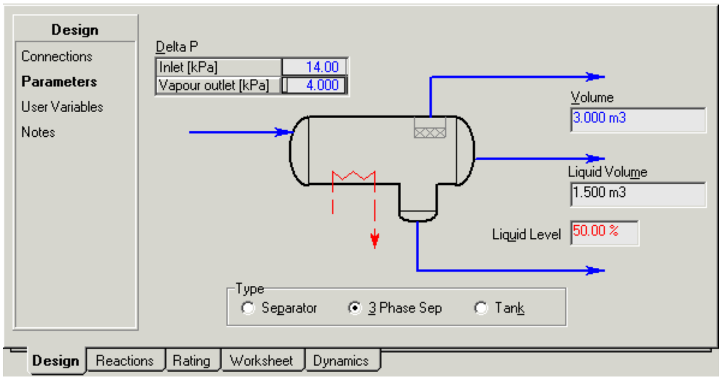


Tank with no energy stream attached.

Parameters Page

The Parameters page allows you to specify the pressure drop across the vessel.

Figure 10.9



The following table lists and describes the options available in this page:

Object	Description
Inlet cell	Allows you to specify the pressure difference across the vessel.
Vapour outlet cell	Allows you to specify the static head pressure of the vessel.
Volume Field	Allows you to specify the volume of the vessel. The default vessel volume is 2 m^3.
Liquid Volume Display Field	Not set by the user. The Liquid Volume is calculated from the product of the Vessel Volume and Liquid Level fraction.
Liquid Level SP Field	Allows you to specify the starting point of the liquid level in the vessel. This value is expressed as a percentage of the Full (Vessel) Volume.
Type Group	Allows you to toggle between the Separator, 3-Phase Separator, and Tank by clicking the appropriate radio button.

If you toggle from a 3 Phase Separator operation to a Separator or Tank operation, you permanently lose the heavy liquid stream connection. If you change back to the 3 Phase Separator, you have to reconnect the heavy liquid stream.

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

> **If you select an alternate unit, your value appears in the face plate using HYSYS display units.**

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

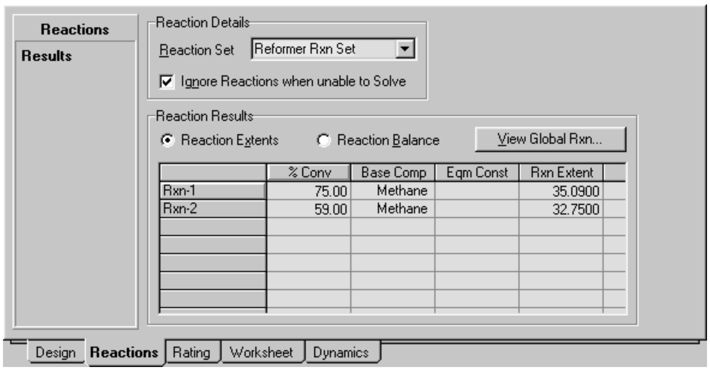
10.2.4 Reactions Tab

The Reactions tab contains options for applying chemical reactions that can take place in the vessel.

Results Page

The Results page allows you to attach a reaction set to the Separator, 3-Phase Separator, or Tank Operations.

Figure 10.10



In the Reaction Set drop-down list, select the reaction set you want to use.

Reaction and component information can also be examined in the Reaction Results group. Select the Reaction Balance radio button to view the total inflow, total reaction, and total outflow for all of the components in the reaction.

Select the Reaction Extents radio button to view the Percent Conversion, Base Component, Equilibrium Constant, and Reaction Extent. You can also view information for specific reactions by clicking the **View Global Rxn** button.

10.2.5 Rating Tab

The Rating tab includes options relevant in both Steady State and Dynamics modes. The options available are:

- Configuring and calculating the separator's vessel size.
- Specifying and calculating heat loss.
- Configuring level taps to observe relative levels of different liquid phases.
- Specifying the PV work term contribution.
- Configuring and calculating the Carry Over model.

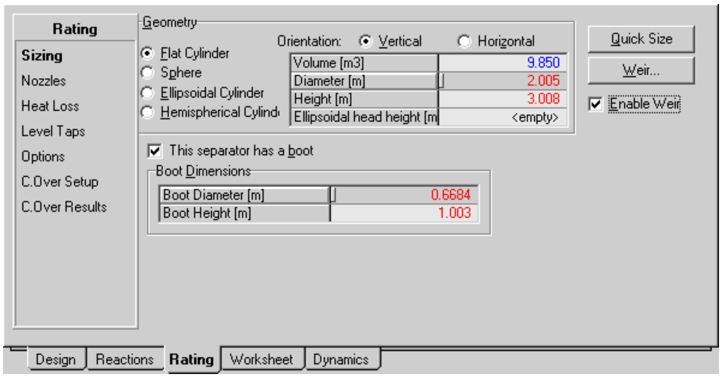
You must provide the following information for the separator operation when working in Dynamics mode:

- vessel geometry
- nozzle geometry
- heat loss

Sizing Page

You can define the geometry of the unit operation on the Sizing page. Also, you can indicate whether or not the unit operation has a boot associated with it. If it does, then you can specify the boot dimensions.

Figure 10.11



After you specified the vessel's dimension information in the appropriate field, click the **Quick Size** button to initiate the HYSYS sizing calculation for the vessel.

Vessel Geometry

In the Geometry group, you can specify the vessel orientation, shape, and volume. The geometry of the vessel is important in determining the liquid height in the vessel.

There are four possible vessel shapes as described in the table below.

Vessel Shape	Description
Flat Cylinder	A cylindrical shape vessel that is available for either horizontal or vertical oriented vessel. You can either specify the total volume or any two of the following for the vessel: • total volume • diameter • height (length) If only the total cylindrical volume of the vessel is specified, the height to diameter ratio is defaulted as 3:2.
Sphere	A sphere shape vessel that is available for either horizontal or vertical oriented vessel. You can either specify the total volume or the diameter of the sphere.

Vessel Shape	Description
Ellipsoidal Cylinder	A ellipsoidal cylindrical shape vessel that is only available for horizontal oriented vessel. You can either specify the total volume or any three of the following for the vessel: • total volume • diameter • length • ellipsoidal head height If only the total cylindrical volume of the vessel is specified, the height to diameter ratio is defaulted as 3:2. If the diameter or length is already specified, the only other variable that can be specified is the Ellipsoidal Head Height.
Hemispherical Cylinder	A hemispherical cylindrical shape vessel that is only available for horizontal oriented vessel. You can either specify the total volume or any two of the following for the vessel: • total volume • diameter • length If only the total cylindrical volume of the vessel is specified, the height to diameter ratio is defaulted as 3:2.

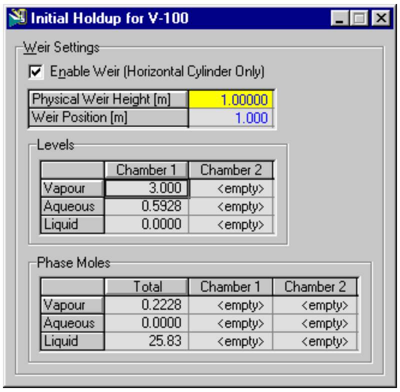
The liquid height in a vertical cylindrical vessel varies linearly with the liquid volume. There is a nonlinear relationship between the liquid height, and the liquid volume in horizontal cylindrical and spherical vessels.

Weir

A weir can be specified for the horizontal flat cylinder separator by selecting the **Enable Weir** checkbox and clicking the **Weir** button. The Initial Holdup property view appears.

The Enable Weir checkbox is only available for Flat Cylinder vessel shape option.

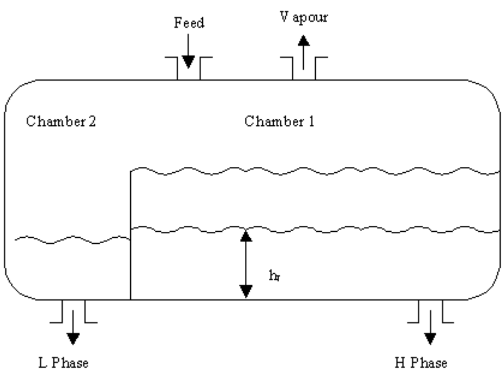
Figure 10.12



The Initial Holdup property view allows you to specify the weir height and position. The weir position is the distance the weir is from the vessel feed side. HYSYS calculates the Levels and Phase Moles in each chamber using the specified values for the weir height and position.

When HYSYS simulates, the weir has two volumes inside the Separator, called chamber 1 and chamber 2; but there is still only one enhanced holdup volume and moles as far as the pressure flow solver is concerned. This means that the compositions and properties of the phases in the two volumes are the same.

Figure 10.13



Boot Geometry

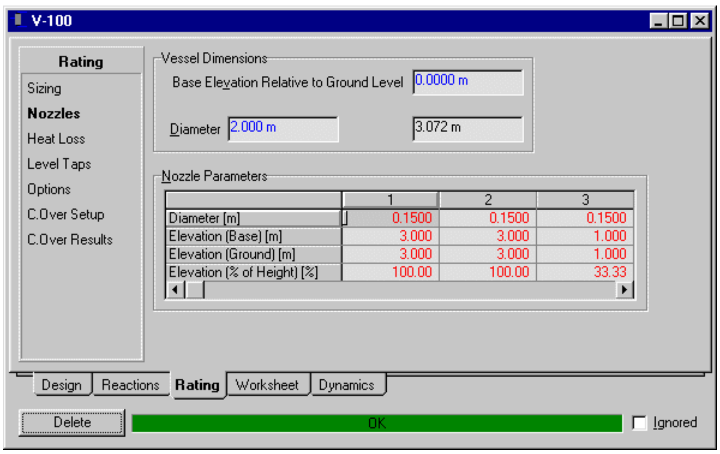
Any vessel operation can be specified with a boot. A boot is typically added when two liquid phases are present in the holdup. Normally, the heavy liquid exits from the boot exit nozzle. The lighter liquid can exit from another exit nozzle attached to the vessel itself.

In HYSYS, a boot can be added to the vessel geometry by selecting the **This Separator has a Boot** checkbox. The boot height is defaulted to one third the vessel height. The boot diameter is defaulted to one third the vessel diameter.

Nozzles Page

The Nozzles page contains information regarding the elevation and diameter of the nozzles.

Figure 10.14



The HYSYS Dynamics license is required to use the Nozzle features found on the Nozzles page.

Unlike steady state vessel operations, the placement of feed and product nozzles on a dynamic vessel operation has physical meaning. The composition of the exit stream depends on the exit stream nozzle location and diameter in relation to the physical holdup level in the vessel.

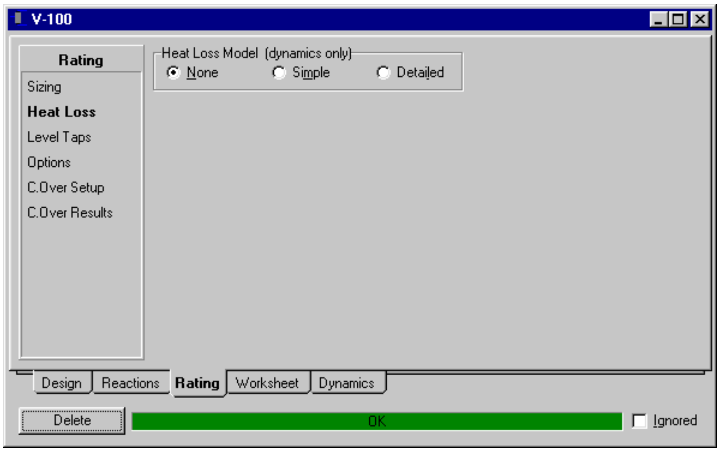
- If the product nozzle is located below the liquid level in the vessel, the exit stream draws material from the liquid holdup.
- If the product nozzle is located above the liquid level, the exit stream draws material from the vapour holdup.
- If the liquid level sits across a nozzle, the mole fraction of liquid in the product stream varies linearly with how far up the liquid is in the nozzle.

Essentially, all vessel operations in HYSYS are treated the same. The compositions and phase fractions of each product stream depend solely on the relative levels of each phase in the holdup and the placement of the product nozzles. So, a vapour product nozzle does not necessarily produce pure vapour. A 3-Phase Separator may not produce two distinct liquid phase products from its product nozzles.

Heat Loss Page

The Heat Loss page contains heat loss parameters which characterize the amount of heat lost across the vessel wall.

Figure 10.15



In the Heat Loss Model group, you can select one of the radio buttons to use as a model:

- Simple
- Detailed
- None (no heat loss through the vessel walls).

The Simple and Detailed heat loss models are discussed in the following sections.

Simple Model

The Simple model allows you to either specify the heat loss directly or have the heat loss calculated from specified values:

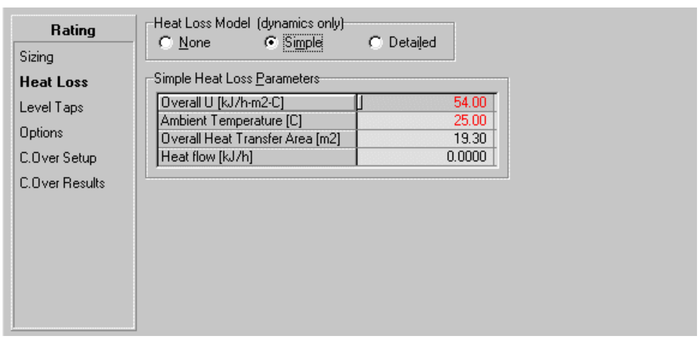
- Overall U value
- Ambient Temperature

The heat transfer area, A, and the fluid temperature, T_f, are calculated by HYSYS. The heat loss is calculated using:

$$Q = UA(T_f - T_{amb}) \tag{10.10}$$

For a Separator, the parameters available for the Simple model are shown in the figure below:

Figure 10.16



The simple heat loss parameters are:

- Overall Heat Transfer Coefficient
- Ambient Temperature

- Overall Heat Transfer Area
- Heat Flow

The Heat Flow is calculated as follows:

$$Heat\ Flow = UA(T_{Amb} - T) \tag{10.11}$$

where:

U = *overall heat transfer coefficient*

A = *heat transfer area*

T_{Amb} = *ambient temperature*

T = *holdup temperature*

As shown, Heat Flow is defined as the heat flowing into the vessel. The heat transfer area is calculated from the vessel geometry. The ambient temperature, T_{Amb}, and overall heat transfer coefficient, U, can be modified from their default values shown in red.

Detailed Model

Refer to **Section 1.6.1 - Detailed Heat Model** in the **HYSYS Dynamic Modeling** guide for more information.

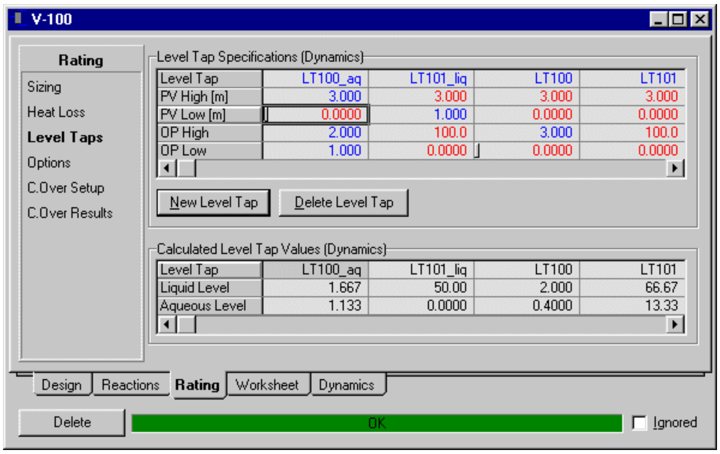
The Detailed model allows you to specify more detailed heat transfer parameters. The HYSYS Dynamics license is required to use the Detailed Heat Loss model found on the Heat Loss page.

Level Taps Page

Since the contents in a vessel can be distributed in different phases, the Level Taps page allows you to monitor the level of liquid and aqueous contents that coexist within a specified zone in a tank or separator.

The information available on this page is relevant only to dynamics cases.

Figure 10.17



Level Taps Specifications (Dynamics)

The Level Tap Specifications (Dynamics) group allows you to specify the boundaries to be monitored within the vessel, and to normalize that section in a desired scale.

A level tap can be specified for any horizontal or vertical vessel by clicking the **New Level Tap** button.

You can add / configure multiple level taps.

To set the boundaries of the section concerned, specify the following fields:

Field	Description
Level Tap	Name of the level tap.
PV High (m)	Upper limit of the section to be monitored. It is expressed in meters.
PV Low (m)	Lower limit of the section to be monitored. It is expressed in meters.
OP High	Upper limit of the output of the normalization scale.
OP Low	Lower limit of the output of the normalization scale.

The normalization scale can be negative values. In some cases, the output normalization scale is manually set between -7% to 100% or -15%-100% so that there is a cushion range before the level of the content becomes unacceptable (in other words, too low or too high).

By default, a new level tap is set to the total height of the vessel, and the height is normalized in percentage (100-0).

All the upper limit specifications should not be smaller than or equal to the lower limit specifications and vice versa; otherwise no calculations will be performed.

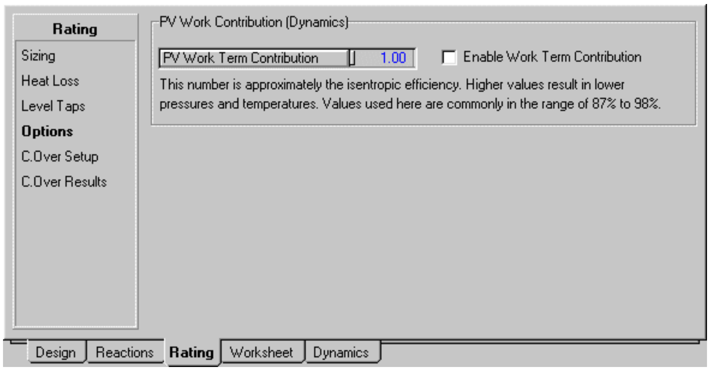
Calculated Level Taps Values (Dynamics)

The level of liquid and aqueous are displayed in terms of the output normalization scale you specified. Whenever the level of a content exceeds PV High, HYSYS automatically outputs the OP High value as the level of that content. If the level is below the PV Low, HYSYS outputs the OP Low value. The levels displayed are always entrained within the normalized zone.

Option Page

The Options page allows you to specify and enable the PV Work Term Contribution.

Figure 10.18



The PV Work Term Contribution is expressed in percent. It is approximately the isentropic efficiency. A high PV work term contribution value results in lower pressures, and temperatures. The PV work term contribution value should be between 87% to 98%.

C.Over Setup Page

The C.Over Setup page allows you to modify the separator operation from an ideal model to a real model.

To achieve a real model separator, the C. Over Setup page enables you to configure the carry over effect in real separator operations. Carry over refers to the conditions when the liquid gets entrained in the vapour phase and/or when the gas gets entrained in the liquid phase. The effect is mainly caused by the disturbances created as the inlet stream enters the vessel.

In HYSYS, the carry over effect is modelled using the entrainment fraction in the feed or product stream, or using the available correlations that calculate carry over based on the vessel configuration.

On the C. Over Setup page, you can select the type of carry over calculation model by clicking one of the following four radio buttons:

- None (indicates that there is currently no carry over model applied to the associated vessel).
- Feed Basis
- Product Basis
- Correlation Based

There are two checkboxes available at the bottom of the page:

- **Carry Over to Zero Flow Streams**. When you select this checkbox, the calculated carry over will be added to the product stream even if it has no flow.
- **Use PH Flash for Product Streams**. When you select this checkbox, you apply a PH flash calculation to the product streams. This option is slower but it may be required to eliminate inconsistencies when one product flow is much less than the others.

These two checkboxes are available in the Feed Basis, Product

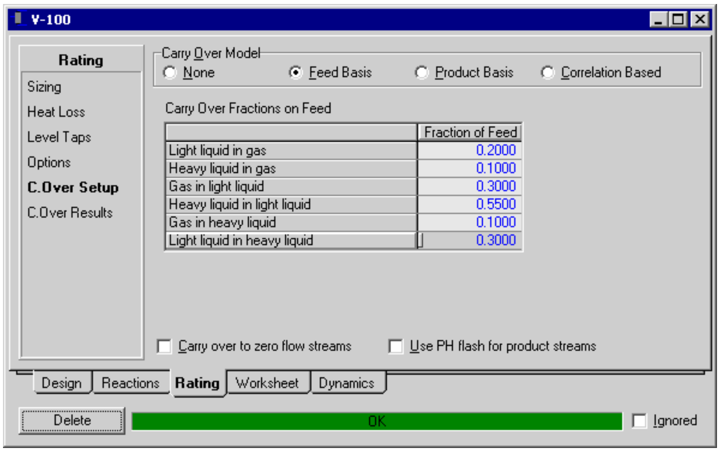
Basis, and Correlation Based models.

The Feed Basis, Product Basis, and Correlation Based models are discussed in the following sections.

Feed Basis Model

The Feed Basis Model allows you to specify the entrainment of each phase in each product as a fraction of the feed flow of the phase.

Figure 10.19



There are six types of carry over flow (in the feed and product streams) available for you to specify:

- Light liquid in gas
- Heavy liquid in gas
- Gas in light liquid
- Heavy liquid in light liquid
- Gas in heavy liquid
- Light liquid in heavy liquid

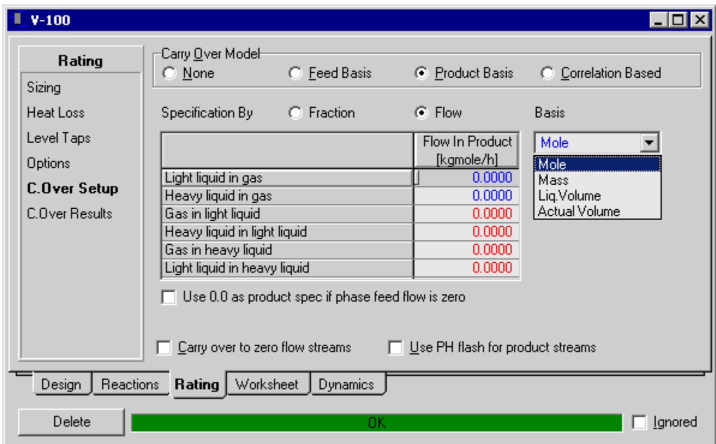
The terms light liquid and heavy liquid refer to oil and water, respectively. No assumptions are made as to the actual composition of the two liquid phases.

The fractions containing non-zero values indicates the product streams exiting the separator will have multiple liquid and gas phases. For example, if you specify Fraction of Feed as 0.1 for light liquid in gas, this means that 10 mol% of the light liquid phase in the feed will be carried over into the gas product leaving the separator. As a result the gas product vapour fraction will be less than 1.0 and contain a liquid phase.

Product Basis Model

The Product Basis model allows you to specify the carry over entrainment in the product streams on fraction or flow basis.

Figure 10.20



You can select the desired basis by clicking on one of the following radio buttons in the Specification By group:

- **Fraction**. Allows you to specify the entrainment in the product stream as a fraction. The fraction basis is selected from the Basis drop-down list and may be either Mole, Mass, Liq.Volume or Actual Volume.
- **Flow**. Allows you to specify the entrainment in the product streams as a flow. The flow basis may be specified using the Flow Basis drop-down list. The options are Mole, Mass, Liq.Volume or Actual volume.

There are six types of carry over flow (in the feed and product streams) available for you to specify:

- Light liquid in gas
- Heavy liquid in gas
- Gas in light liquid
- Heavy liquid in light liquid
- Gas in heavy liquid
- Light liquid in heavy liquid

The terms light liquid and heavy liquid refer to oil and water, respectively. No assumptions are made as to the actual composition of the two liquid phases.

For example, if you specify Frac in Product (Mole Basis) as 0.1 for the light liquid in gas, this means that the gas product will contain 10 mol% light liquid.

In Steady State mode, if a phase is missing from the feed stream, selecting the **Use 0.0 as product spec if phase feed flow is zero** allows the separator to continue to calculate the carry over effect (in this example, carry over model calculation ignores any product fraction or flow specification for that phase).

In Dynamics mode, the **Use 0.0 as product spec if phase feed flow is zero** feature is automatically and always active.

Correlation Based Model

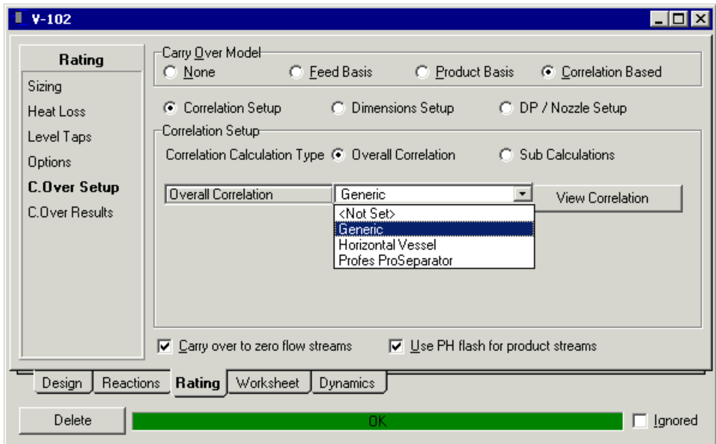
The Correlation Based model allows you to calculate the expected carry over based on the configuration of the vessel, the feed conditions, and the operating conditions.

The options available in the Correlation Based model allows you to configure the type of the correlation, dimensions and geometry of vessel, pressure drop methods, and nozzle location.

Correlation Setup

The Correlation Setup group allows you to select the Correlation Calculation Type, and how you want to apply the correlation.

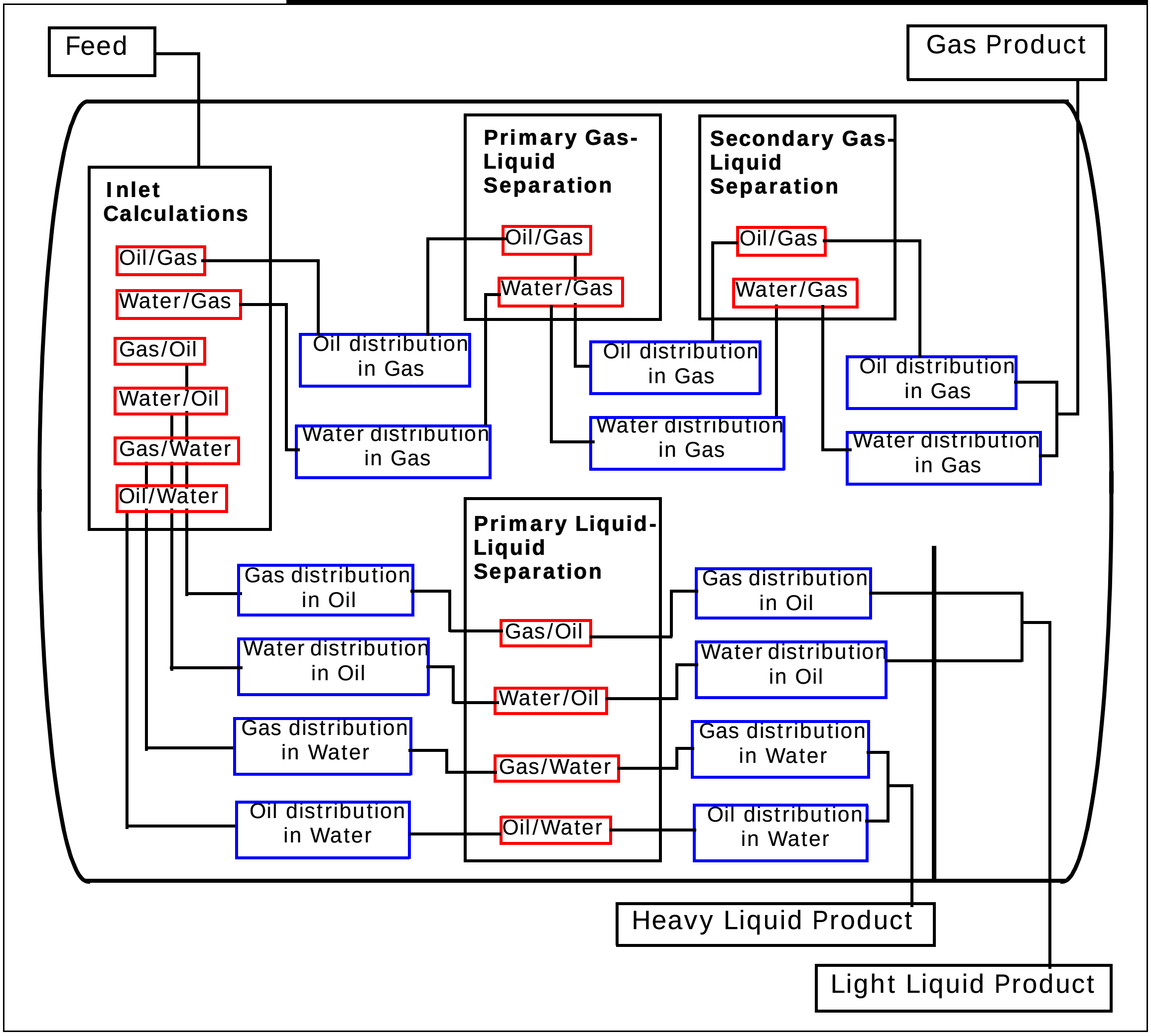
Figure 10.21



- You can apply one correlation for all of the carry over calculations by clicking on the **Overall Correlation** radio button.
- If you want to select a different correlation for each carry over calculation steps (Inlet Device, Gas/Liq Separation, Liq/Liq Separation, and Vapour Exit Device), you can select the **Sub Calculations** radio button to activate the appropriate options.

A schematic of these steps is shown in the figure below:

Figure 10.22



Only those parts of the correlation in use that apply to the particular sub-calculation will be used.

For 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 (in other words, 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.

There are three correlations available: Generic, Horizontal Vessel, and ProSeparator. After you have selected the type of correlation, you can click on the **View Correlation** button to view its calculation parameters.

- Generic

  The Generic correlation provides a general correlation for generating the phase dispersions in the feed and defining the separation criteria. It is a generic calculation that is independent of the vessel dimensions, geometry, and operating levels.

  For the Inlet Calculations, you must define the percentage of each feed phase dispersed in each other phase. You must also define the maximum droplet sizes and Rosin Rammler index for each dispersion. The dispersions are then calculated using Rosin Rammler methods to give the amount of each phase in each droplet size range.

  For the rest of the carry over calculation, all droplets smaller than a user-specified critical droplet size are assumed to be carried over.

  The Generic correlation can be used for gas-liquid, liquid-liquid, and gas exit separation calculations. You must define the critical droplet size for each type of separation. Any droplets that are smaller than the specified critical droplet size will be carried over; the droplets that are larger than the critical droplet size will be separated.

- Horizontal Vessel[1 2 3]

  The Horizontal Vessel correlations were developed for a horizontal three-phase separator.

  For the Inlet Calculations, the correlations calculate the six types of dispersions in the feed according to an assumed efficiency of a user-defined inlet device, and user-defined dispersion fractions (termed **Inlet Hold up**; these parameters are found on the **General** page in the **Setup** tab of the Horizontal Vessel Correlation property view). The droplet distribution of the dispersed phase(s) is then calculated using user-supplied Rosin-Rammler parameters just as for the Generic correlation.

The droplet d95 of the liquid-liquid dispersions (in other words, heavy liquid in light liquid and light liquid in heavy liquid) is not specified but calculated using the inlet droplet d95 and the densities of the 2 liquid phases.

The Primary Gas-Liquid Separation is calculated from the settling velocities for each liquid (light and heavy) droplet size in the gas phase and the residence time for the gas in the vessel. A droplet is carried over if the vertical distance travelled during its residence in the vessel is less than the vertical distance required to rejoin its bulk phase.

The Primary Liquid-Liquid Separation is also calculated using settling velocities for each droplet of liquid or gas in the liquid phases and residence time for each liquid phase. The settling velocities are calculated using the GPSA correlations for all dispersions, except for the water in oil dispersion for which the settling velocity is calculated by the method of Barnea and Mizrahi. A user defined liquid phase inversion point is used in the calculation of the appropriate liquid phase viscosities (in other words, water-in-oil and oil-in-water). A residence time correction factor can also be applied. A droplet is carried over if the vertical distance travelled during its residence in the vessel is less than the vertical distance required to rejoin its bulk phase.

The Secondary Separation calculations for the gas phase are defined by a user-defined critical droplet size. The gas loading factor for each device is used to calculate the size of the exit device.

- ProSeparator[4]

  The ProSeparator correlations are rigorous but are limited to calculating liquid carry over into gas. There are no calculations of liquid-liquid separation or gas entrainment in the liquid phases (they are set to zero). Light liquid and heavy liquid entrainments are calculated separately and the total carry over is the sum of the separate light and heavy liquid carry over calculations.

  For Inlet Calculations, minimum and maximum droplet diameter are calculated based on inlet flow conditions (inlet gas flow rate and gas-liquid phase physical properties) and inlet pipe size. The droplet distribution of light and heavy liquids in the inlet gas is then calculated using a Rosin-Rammler type distribution.

ProSeparator effectively calculates its own Rossin Rammler parameters[5] (droplet diameters) and does not require the user to specify these parameters. The only user input in the inlet calculations is the ability to limit the amount of phase dispersion calculated.

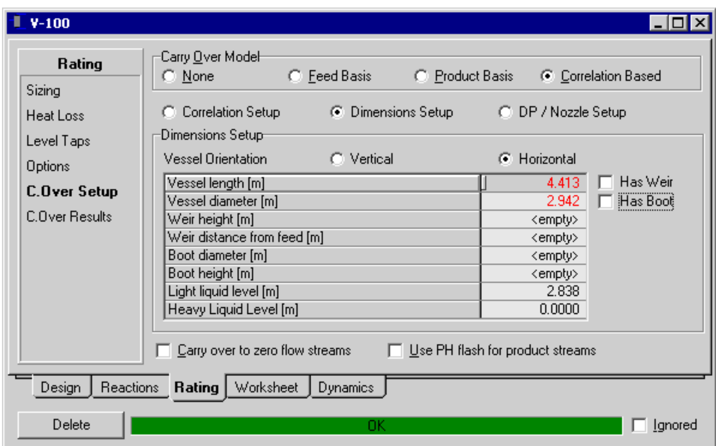
Primary Gas-Liquid Separation is based on critical droplet size; however, the critical droplet size is not user-specified but calculated using gas velocity through the vessel.

Secondary Gas-Liquid Separation accomplished using exit devices (for example, demisting pad) are calculated by device specific correlations. The user can choose from vane pack or mesh pad devices. There are 2 different calculation methods available for each type of device.

Dimensions Setup

The Dimensions Setup group allows you to set the orientation, and the geometry of the vessel. You can also set the operating levels for the light and heavy liquid in the vessel. You have the option to model the horizontal vessel with weir or a boot by selecting the **Has Weir** checkbox and **Has Boot** checkbox, respectively.

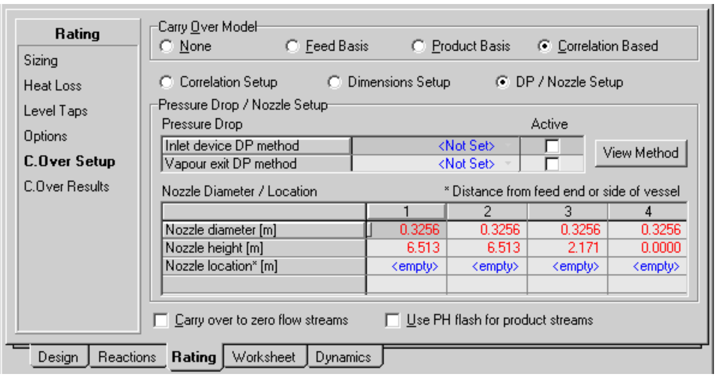
Figure 10.23

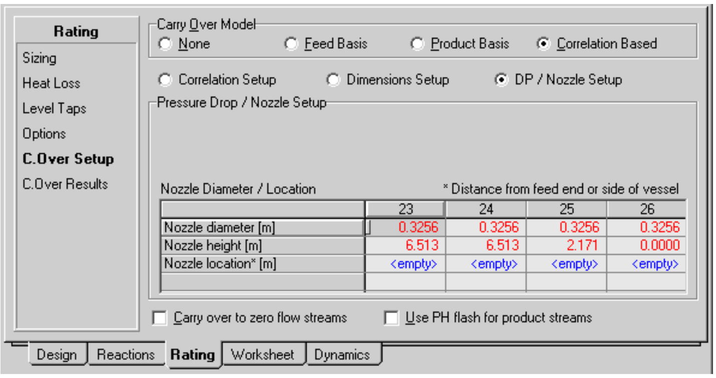


DP/Nozzle Setup

The Pressure Drop/Nozzle Setup group allows you to model the method of DP (pressure drop), and the geometry of the nozzles.

Figure 10.24





Since the dynamic pressure and the pressure drop of the feed stream and vessel are specified in the Specs page of the Dynamics tab, the Pressure Drop table is not available when the separator is operating in dynamics mode.

In the Pressure Drop table, after you have selected a DP method from the drop-down list, you can activate the Inlet Device DP Method, and the Vapour Exit DP Method by selecting the **Active** checkbox. You can click on the **View Method** button to view the parameters of the DP method you have selected.

In the Nozzle Diameter/Location table, you can set the diameter, height, and location for all the streams connected to the vessel.

If a Correlation Based carry over model is selected:

- the options on the **DP / Nozzles Setup** page can be used to calculate the inlet and exit nozzle pressure drop.
- the user-specified pressure drops on the **Parameters** page of the **Design** tab can be used instead.

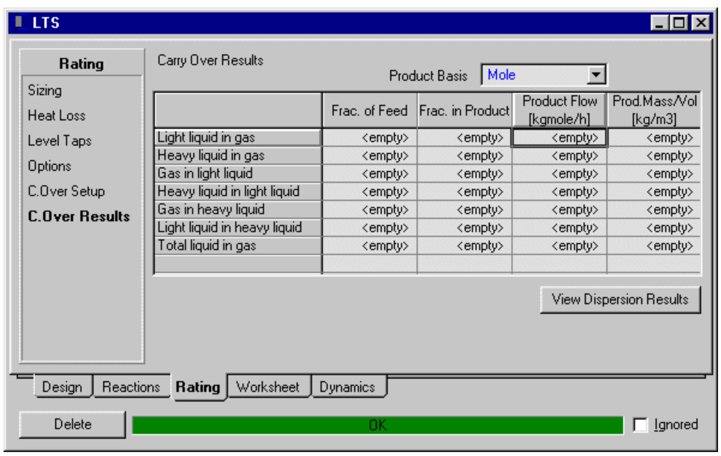
C. Over Results Page

The C. Over Results page allows you to view the carry over results in the feed, and product streams based on what you specified in the C. Over Setup page. There are four columns of data in the Carry Over Results table:

- Frac. of Feed
- Frac. in Product
- Product Flow
- Prod. Mass/Vol

The C. Over Results page information is also available in Dynamic mode.

Figure 10.25



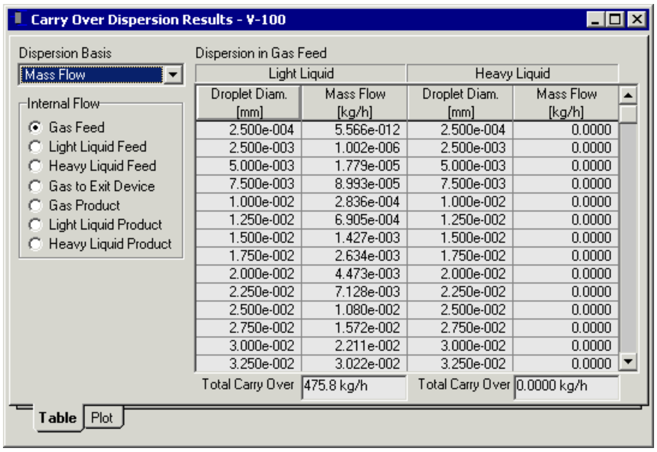
The units for Frac. of Feed, and Prod. Mass/Vol are set by default. You can change the unit for the Frac. in Product and Product Flow column by selecting one of the four units from the Product Basis drop-down list (Mole, Mass, Liq. Volume, and Actual Volume).

You can view the carry over dispersion results by clicking on the **View Dispersion Results** button.

The Carry Over Dispersion Results property view has two tabs:

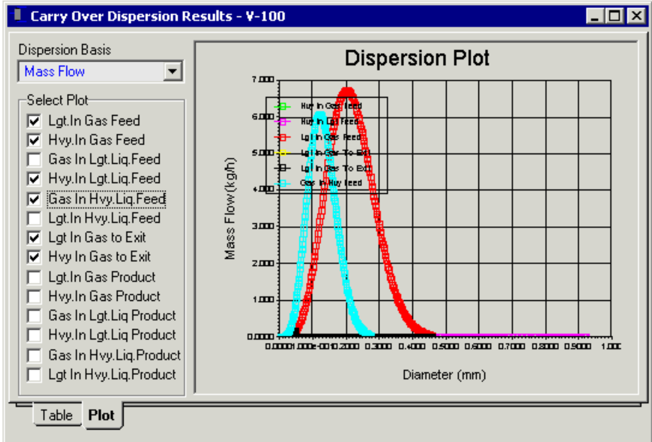
- **Table**. Displays the dispersion results for a single phase at a given point in the vessel. The radio buttons allow you to select the results of the corresponding phase. You can select the unit to be displayed from the Dispersion Basis drop-down list.

Figure 10.26



- **Plot**. Provides a graphically interpretation of the dispersed quantity against the droplet size for a single dispersion. The Select Plot checkboxes allow you to select one or more dispersions to be plotted. You can select the dispersion basis from the Dispersion Basis drop-down list.

Figure 10.27



10.2.6 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

The PF Specs page is relevant to dynamics cases only.

10.2.7 Dynamics Tab

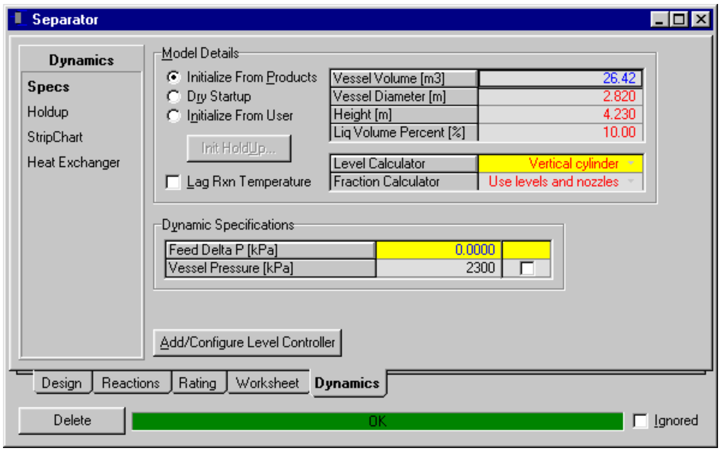
Most of the options available on this tab is relevant only to cases in Dynamics mode.

There is one exception, the **Heat Exchanger Page** allows you to specify whether or not the separator operation contains an energy stream used to heat or cool the vessel.

Specs Page

The Specs page contains information regarding initialization modes, vessel geometry, and vessel dynamic specifications.

Figure 10.28



Model Details

You can determine the composition and amount of each phase in the vessel holdup by specifying different initialization modes. HYSYS forces the simulation case to re-initialize whenever the initialization mode is changed. The radio buttons in the Model Details group are briefly described in the table below.

Initialization Mode	Description
Initialize from Products	The composition of the holdup is calculated from a weighted average of all products exiting the holdup. A PT flash is performed to determine other holdup conditions. The liquid level is set to the value indicated in the Liq Volume Percent field.

Initialization Mode	Description
Dry Startup	The composition of the holdup is calculated from a weighted average of all feeds entering the holdup. A PT flash is performed to determine other holdup conditions. The liquid level in the Liq Volume Percent field is set to zero.
Initialize from User	The composition of the liquid holdup in the vessel is user specified. The molar composition of the liquid holdup can be specified by clicking the Init Holdup button. The liquid level is set to the value indicated in the Liq Volume Percent field.

In the Model Details group, you can specify the vessel geometry parameters:

The vessel geometry parameters can be specified in the same manner as those specified in the Geometry group for the Sizing page of the Rating tab.

- Vessel Volume
- Vessel Diameter
- Vessel Height (Length)
- Liq Volume Percent

  You can modify the level in the vessel at any time. HYSYS then uses that level as an initial value when the integrator is run.
- Vessel Geometry (Level Calculator)
- Fraction Calculator

Fraction Calculator

The Fraction Calculator defaults to the correct mode for all unit operations and does not typically require any changing.

The Fraction Calculator determines how the levels in the tank, and the elevation and diameter of the nozzles affect the product composition. The following is a description of the Fraction Calculator options:

For more information, see the section on **Nozzles Page**.

- **Use Levels and Nozzles.** The nozzle location and vessel phase (liquid/vapour) level determines how much of each phase, inside the vessel, will exit through that nozzle.

  For example, if a vessel contained both liquid and vapour phases and the nozzle is below the liquid level, then liquid will flow out through it. If the nozzle is above the liquid level, then vapour will flow out through it.

- **Emulsion Liquids.** This behaves like the Use Levels and Nozzles option, except it simulates a mixer inside the vessel that mixes two liquid phases together so they do not separate out.

  For example, if a nozzle is below the lighter liquid level and the vessel has two liquid phases, the product is a mixture of both liquid phases.

Dynamic Specifications

The frictional pressure loss at the feed nozzle is a dynamic specification in HYSYS. It can be specified in the Feed Delta P field. The frictional pressure losses at each product nozzle are automatically set to zero by HYSYS.

It is recommended that you enter a value of zero in the Feed Delta P field because a fixed pressure drop in the vessel is not realistic for all flows.

If you want to model friction loss at the inlet and exit stream, it is suggested you add valve operations. In this case, flow into and out of the vessel is realistically modeled.

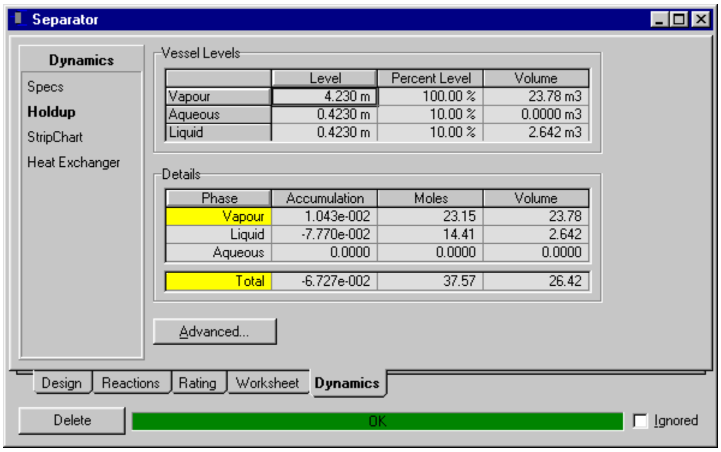
The vessel pressure can also be specified. This specification can be made active by selecting the checkbox beside the **Vessel Pressure** field. This specification is typically not set since the pressure of the vessel is usually a variable and determined from the surrounding pieces of equipment.

Holdup Page

Refer to **Section 1.3.3 - Holdup Page** for more information.

The Holdup page contains information regarding the properties, composition, and amount of the holdup.

Figure 10.29



The Vessel Levels group displays the following variables for each of the phases available in the vessel:

- Level. Height location of the phase in the vessel.
- Percent Level. Percentage value location of the phase in the vessel.
- Volume. Amount of space occupied by the phase in the vessel.

Stripchart Page

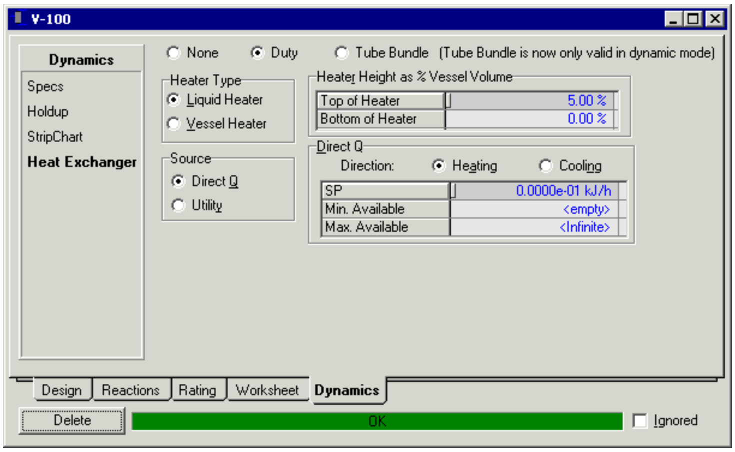
Refer to **Section 1.3.7 - Stripchart Page / Tab** for more information.

The Stripchart page allows you to select a default strip chart containing various variable associated to the operation.

Heat Exchanger Page

The Heat Exchanger page allows you to select whether the unit operation is heated, cooled, or left alone. You can also select the method used to heat or cool the unit operation.

Figure 10.30



The options available in the **Heat Exchanger** page depends on which radio button you select:

Refer to **Duty Radio Button** for more information.

Refer to **Tube Bundle Radio Button** for more information.

- If you select the **None** radio button, this page is blank and you do not have to specify an energy stream in the **Connections** page (from the **Design** tab) for the separator operation to solve.
- If you select the **Duty** radio button, this page contains the standard heater or cooler parameters and you have to specify an energy stream in the **Connections** page (from the **Design** tab) for the separator operation to solve.
- If you select the **Tube Bundle** radio button, this page contains the parameters used to configure a heat exchanger and you have to specify material streams in the **Connections** page (from the **Design** tab) for the separator operation to solve.

The Tube Bundle options are only available in Dynamics mode and for Separator and Three Phase Separator.

If you switch from Duty option or Tube Bundle option to None option, HYSYS automatically disconnects the energy or material streams associated to the Duty or Tube Bundle options.

10.3 Shortcut Column

The Shortcut Column performs Fenske-Underwood short cut calculations for simple refluxed towers. The Fenske minimum number of trays and the Underwood minimum reflux are calculated. A specified reflux ratio can then be used to calculate the vapour and liquid traffic rates in the enriching and stripping sections, the condenser duty and reboiler duty, the number of ideal trays, and the optimal feed location.

The Shortcut Column is only an estimate of the Column performance and is restricted to simple refluxed Columns. For more realistic results the rigorous Column operation should be used. This operation can provide initial estimates for most simple Columns.

10.3.1 Shortcut Column Property View

There are two ways that you can add a Shortcut Column to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

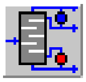
   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Short Cut Columns** radio button.
3. From the list of available unit operations, select the **Shortcut Column** model.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.

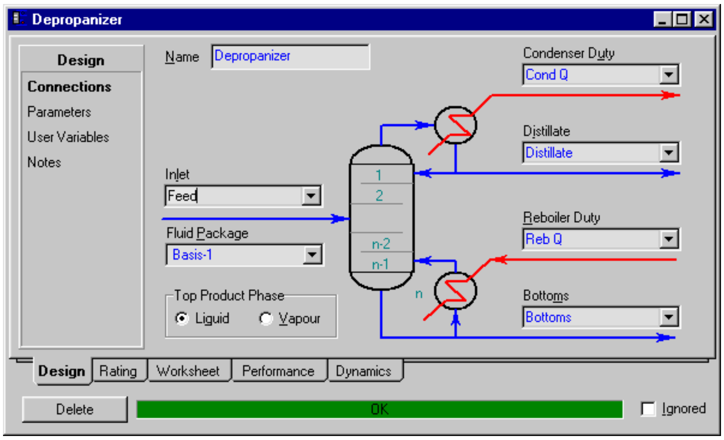
2. Double-click the **Short Cut Distillation** icon.



Short Cut Distillation icon

The Shortcut Column property view appears.

Figure 10.31



10.3.2 Design Tab

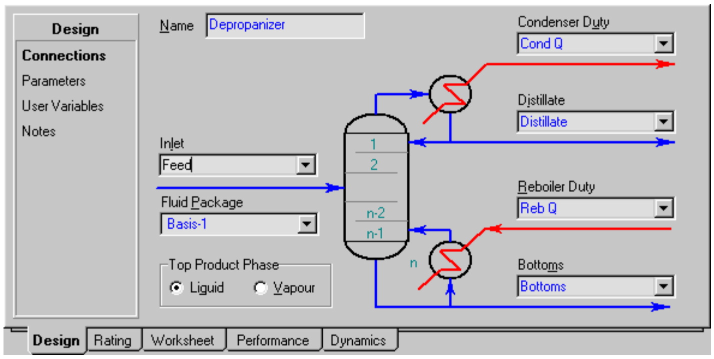
The Design tab contains the following pages:

- Connections
- Parameters
- User Variables
- Notes

Connections Page

You must specify the feed stream, overhead product, bottoms product, condenser, and reboiler duty name on the Connections page.

Figure 10.32



The overhead product can either be an overhead vapour or a distillate stream, depending on the radio button selected in the Top Product Phase group. The operation name can also be changed on this page.

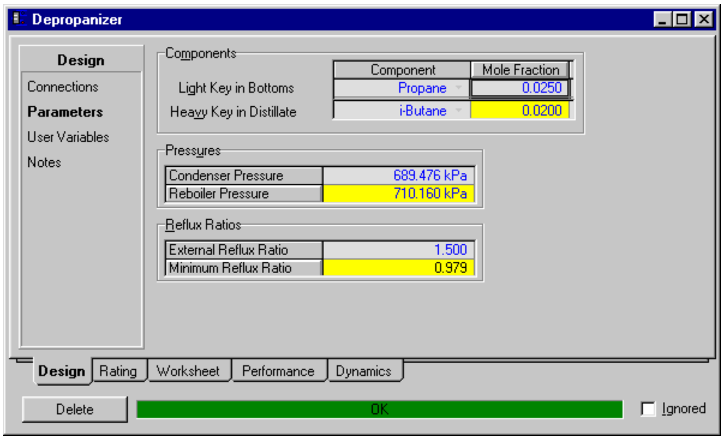
You can specify the top product to be either liquid or vapour using the radio buttons in the Top Product Phase group.

Parameters Page

The Shortcut Column requires the light key and heavy key components to be defined. The light key is the more volatile component of the two main components that are to be separated. The compositions of the keys are used to specify the distillation products.

The composition of the light key in the bottoms and the heavy key in the overhead are the only composition specifications required.

Figure 10.33



In the Components group, select the light key in bottoms and heavy key in distillation component from the drop-down list in the component cell, and specify their corresponding mole fraction. The specification must be such that there is enough of both keys to be distributed in the bottoms and overhead. It is possible to specify a large value for the light key composition such that too much of the light key is in the bottoms, and the overhead heavy key composition spec cannot be met. If this problem occurs, one or both of the key specs must be changed.

In the Pressures group, you can define the column pressure profile by specifying a value in the Condenser Pressure field and a Reboiler Pressure field.

In the Reflux Ratios group, the calculated minimum reflux ratio appears once streams are attached on the Connections page, and the required parameters are specified in the Components group and Pressures group.

You can then specify an external reflux ratio, which is used to calculate the tray traffic, the condenser and reboiler duties, the ideal number of trays, and the optimum tray location. The external reflux ratio must be greater than the minimum reflux ratio.

10.3.3 Rating Tab

You currently cannot provide any rating information for the Shortcut Column.

10.4 Worksheet Tab

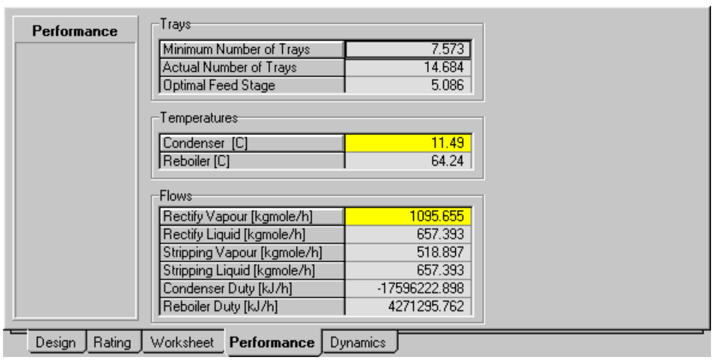
Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab displays a summary of the information contained in the stream property view for all the streams attached to the operation.

10.3.5 Performance Tab

The Performance tab allows you to examine the results of the Shortcut Column calculations. The results correspond to the external reflux ratio value that you specified on the Parameters page.

Figure 10.34



The following results are available on the:

Column Result	Description
Minimum Number of Trays	The Fenske minimum number of trays, which is not affected by the external reflux ratio specification.
Actual Number of Trays	Calculated using a using the Gilliland method.

Column Result	Description
Optimal Feed Stage	Top down feed stage for optimal separation.
Condenser and Reboiler Temperatures	These temperatures are not affected by the external reflux ratio specification.
Rectifying Section Vapour and Liquid traffic flow rates	The estimated average flow rates above the feed location.
Stripping Section Vapour and Liquid traffic flow rates	The estimated average flow rates below the feed location.
Condenser and Reboiler Duties	The duties, as calculated by HYSYS.

10.3.6 Dynamics Tab

The Shortcut Column currently runs only in Steady State mode. As such, there is no information available on the Dynamics tab.

10.4 References

[1] GPSA, Vol 1, 10th Ed., January 1990.

[2] Separation Mechanism of Liquid-Liquid Dispersions in a deep-layer Gravity, Settler, E. Barnea and J Mizrahi, Trans. Instn. Chem. Engrs, 1975, Vol 53.

[3] Droplet size spectra generated in turbulent pipe flow of dilute liquid-liquid dispersions, A J Karabelas, AIChE, 1978, vol. 24, No. 2, pages 170-181.

[4] Society of Petroleum Engineers papers:
SPE36647 – Separator Design and Operation: Tools for Transferring "Best Practise"
SPE21506 – Proseparator – a novel separator/scrubber design program

[5] Aspen Process Manuals – Mini Manual 1: Gas & Particle Properties; Part 8 – Particle Size

[6] Aspen Process Manuals – Gas Cleaning Manual:
Vol 1 – Introduction
Vol 2 – Demisting
Vol 10 – Applied Technology

11 Solid Separation Operations

11.1 Baghouse Filter

The Baghouse Filter model is based on empirical equations. It contains an internal curve relating separation efficiency to particle size. Based on your particle diameter, the reported separation efficiency for your solids is determined from this curve. The solids being separated must be previously specified and installed as components in the stream attached to this operation.

11.1.1 Baghouse Filter Property View

There are two ways that you can add a Baghouse Filter to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Solids Handling** radio button.
3. From the list of available unit operations, select **Baghouse Filter**.
4. Click the **Add** button.

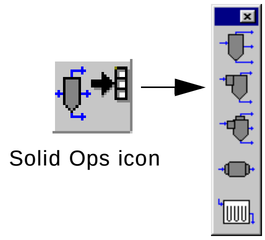
OR

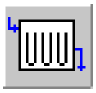
1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.

2. Click on the **Solid Ops** icon. The Solid Operations Palette appears.

Figure 11.1



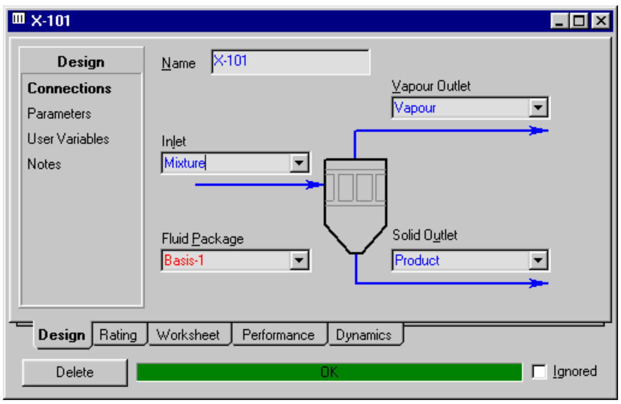


Baghouse Filter icon

3. Double-click the **Baghouse Filte**r icon.

The Baghouse Filter property view appears.

Figure 11.2



11.1.2 Design Tab

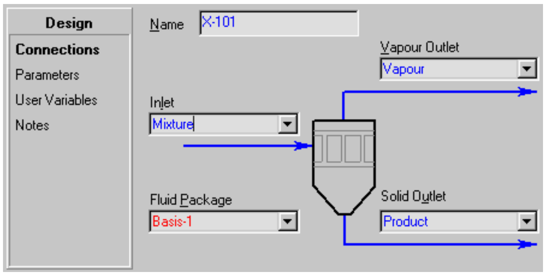
The Design tab contains the following pages:

- Connections
- Parameters
- User Variables
- Notes

Connections Page

On the Connections page, you can specify the name of the operation, as well as the feed, vapour product, and solid product streams.

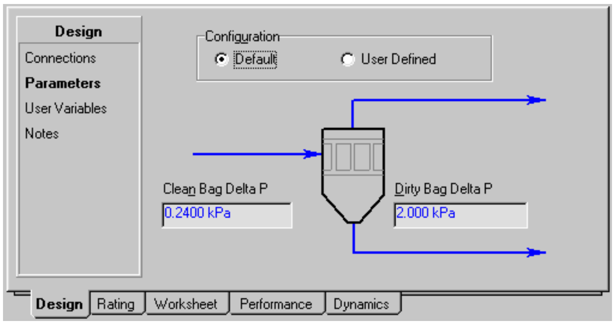
Figure 11.3



Parameters Page

On the Parameters page, you must specify the following information:

Figure 11.4



Parameter	Description
Configuration	When you make a change to any of the parameters, the configuration changes to User Defined. Select Default to revert to the HYSYS defaults.

Parameter	Description
Clean Bag Pressure Drop	The pressure drop across a clean bag.
Dirty Bag Pressure Drop	The pressure drop across a dirty bag. This value must be greater than the Clean Bag Pressure Drop.

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

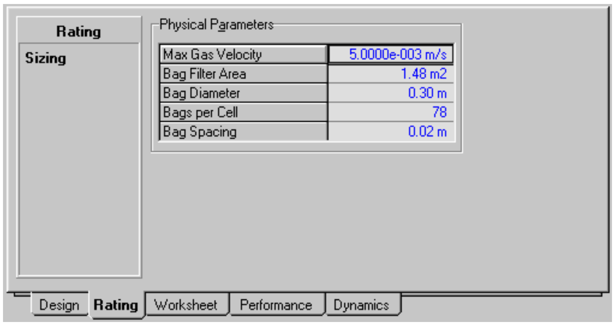
11.1.3 Rating Tab

The Rating tab consists of the Sizing page.

Sizing Page

On the Sizing page, the following parameters can be specified:

Figure 11.5



Parameter	Description
Maximum Gas Velocity	Maximum velocity of gas in the Baghouse Filter.
Bag Filter Area	Filter Area for each bag.
Bag Diameter	Bag Diameter.
Bags per Cell	Number of bags per filter cell.
Bag Spacing	Spacing between the bags.

11.1.4 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

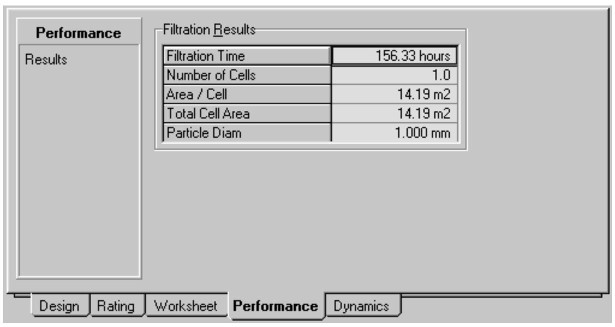
The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

11.1.5 Performance Tab

The Performance tab consists of the Results page.

Results page

Figure 11.6



The following Filtration results appear on this page:

- Filtration Time
- Number of Cells
- Area/Cell
- Total Cell Area
- Particle Diameter

11.1.6 Dynamics Tab

This unit operation is currently not available for dynamic simulation.

11.2 Cyclone

The Cyclone is used to separate solids from a gas stream and is recommended only for particle sizes greater than 5 microns. The Cyclone consists of a vertical cylinder with a conical bottom, a rectangular inlet near the top, and an outlet for solids at the bottom of the cone. It is the centrifugal force developed in the vortex which moves the particles toward the wall. Particles which reach the wall, slide down the cone, and so become separated from the gas stream. The solids being separated must be previously specified and installed as components in the stream attached to this operation.

11.2.1 Cyclone Property View

There are two ways that you can add a Cyclone to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Solids Handling** radio button.
3. From the list of available unit operations, select **Cyclone**.
4. Click the **Add** button.

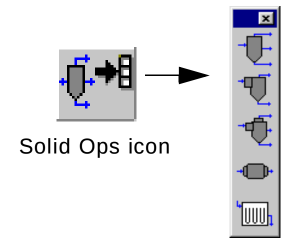
OR

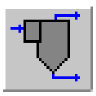
1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.

2. Click on the **Solid Ops** icon. The Solid Operations Palette appears.

Figure 11.7



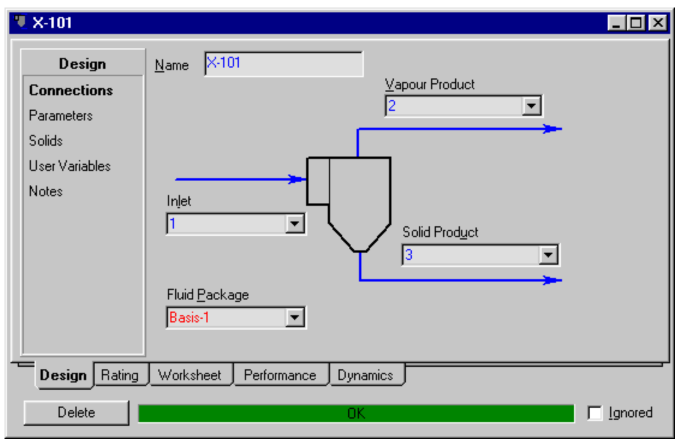


Cyclone icon

3. Double-click the **Cyclone** icon.

The Cyclone property view appears.

Figure 11.8



11.2.2 Design Tab

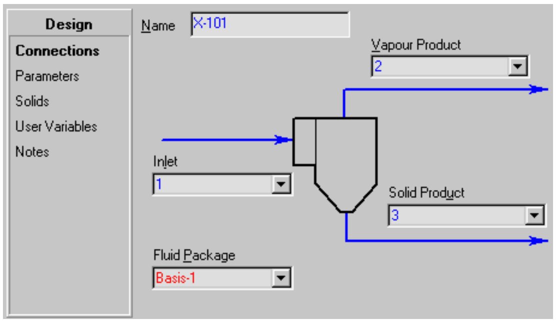
The Design tab contains the following pages:

- Connections
- Parameters
- Solids
- User Variables
- Notes

Connections Page

You can specify the name of the operation, as well as the feed, vapour product, and solid product streams on the Connections page.

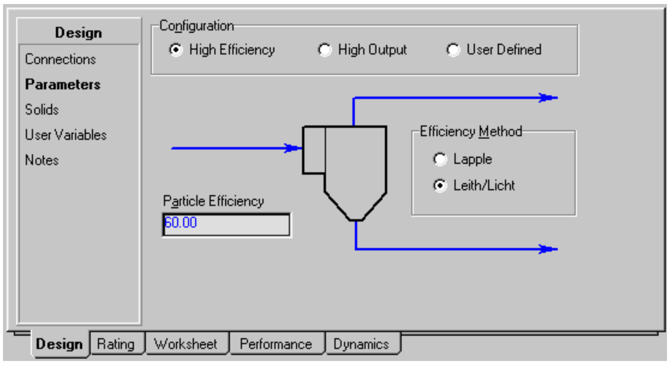
Figure 11.9



Parameters Page

On the Parameters page, you can specify the following parameters:

Figure 11.10



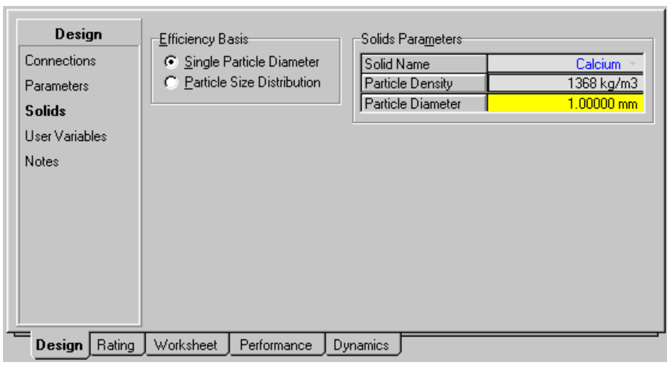
Parameter	Description
Configuration	Select either High Efficiency, High Output or User Defined.
Efficiency Method	Select either the Lapple or the Leith/Licht method. The latter is a more rigorous calculation as it considers radial mixing effects.
Particle Efficiency	The percent recovery of the specified solid in the bottoms stream.

The diameter provided, either from the selected component or from the particle characteristics, is used in the efficiency calculations. For example, if you select an 85% efficiency, 85% of the solids of the specified diameter is recovered. All other solids in the inlet stream are removed at an efficiency related to these parameters.

Solids Page

You can specify the solid characteristics on the Solids page. This page contains two different data, depending on the radio button you selected in the Efficiency Basis group.

Figure 11.11



When you select the Single Particle Diameter radio button, the following solids information can be specified:

Parameter	Description
Solid Name	You must provide either the name of a solid already installed in the case, or provide a particle diameter and density.
Particle Diameter and Particle Density	If you do not choose a solid component, provide the particle diameter and density.

When you select the Particle Size Distribution radio button, the following solids information can be specified:

Parameter	Description
Solid Name	You must provide either the name of a solid already installed in the case, or provide a particle diameter and density.
Particle Density	If you do not choose a solid component, provide the particle density.
Particle Size Distribution	If you do not choose a solid component, provide the minimum or maximum particle size.

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

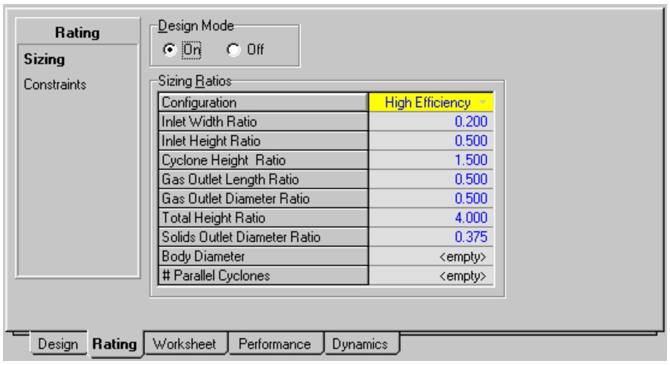
11.2.3 Rating tab

The Rating tab contains the following pages:

- Sizing
- Constraints

Sizing Page

Figure 11.12



The Sizing page contains two groups:

- **Design Mode**. Contains two radio buttons: On and Off.

  The radio buttons enables you to toggle between turning on or turning off the Design Mode option.

  When you select the **Off** radio button, the **Specify Number of Parallel Cyclones** checkbox appears in the Sizing page. Select the checkbox if you want to specify the number of parallel Cyclones in the flowsheet.
- **Sizing Ratios**. Contains a table.

  The table below describes the parameters available on the page:

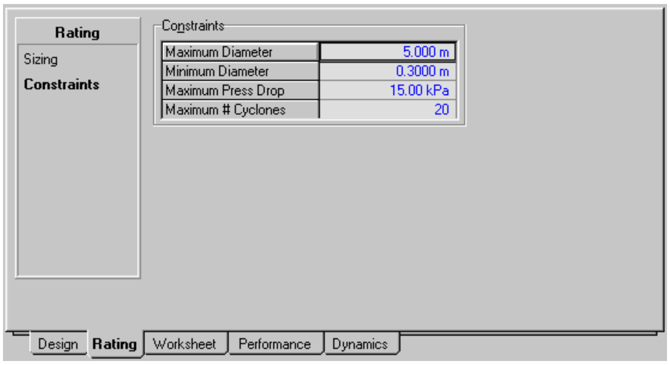
Parameter	Description
Configuration	Select High Output, High Efficiency, or User Defined. This is also defined on the Parameters page.
Inlet Width Ratio	The ratio of the inlet width to the body diameter (must be between 0 and 1). The value must be less than the Total Height Ratio
Inlet Height Ratio	The ratio of the inlet height to the body diameter.
Cyclone Height Ratio	The ratio of the Cyclone height to the body diameter. The Cyclone is the conical section at the bottom of the entire operation.
Gas Outlet Length Ratio	The ratio of the gas outlet length to the body diameter.
Gas Outlet Diameter Ratio	The ratio of the gas outlet diameter to the body diameter (must be between 0 and 1). The value must be less than the Total Height Ratio

Parameter	Description
Total Height Ratio	The ratio of the total height of the apparatus to the body diameter.
Solids Outlet Diameter Ratio	The ratio of the solids outlet diameter to the body diameter.
Body Diameter	If Design Mode is on, this is automatically calculated, within the provided constraints. If Design Mode is off, then you can specify this value.
# Parallel Cyclones	If Design Mode is on, this field displays the number of parallel Cyclones (if any) attached to the unit operation. If Design Mode is off and the **Specify Number of Parallel Cyclones** checkbox is selected, you can specify this value.

Constraints Page

On the Constraints page, you can specify the minimum and maximum diameter for the Cyclone. The page is also applicable only when the On radio button is selected in the Design Mode group.

Figure 11.13



The Maximum Pressure Drop and Maximum Number of Cyclones is set on this page. These are used in the calculations to determine the minimum number of Cyclones needed to complete the separation.

11.2.4 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

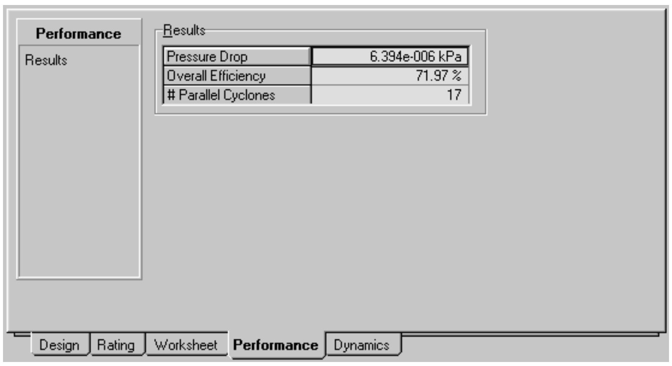
11.2.5 Performance Tab

The Performance tab contains the Results page.

Results Page

The Results page displays the calculated pressure drop, overall efficiency, and the number of parallel Cyclones.

Figure 11.14



11.2.6 Dynamics Tab

This unit operation is currently not available for dynamic simulation.

11.3 Hydrocyclone

The Hydrocyclone is essentially the same as the cyclone, the primary difference being that this operation separates the solid from a liquid phase, rather than a gas phase. The solids being separated must be previously specified and installed as components in the stream attached to this operation.

11.3.1 Hydrocyclone Property View

There are two ways that you can add a Hydrocyclone to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitsOps property view by pressing **F12**.
2. Click the **Solids Handling** radio button.
3. From the list of available unit operations, select **Hydrocyclone**.
4. Click the **Add** button.

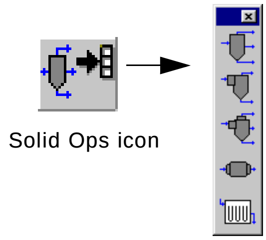
OR

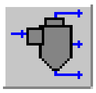
1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.

2. Click on the **Solid Ops** icon. The Solid Operations Palette appears.

Figure 11.15



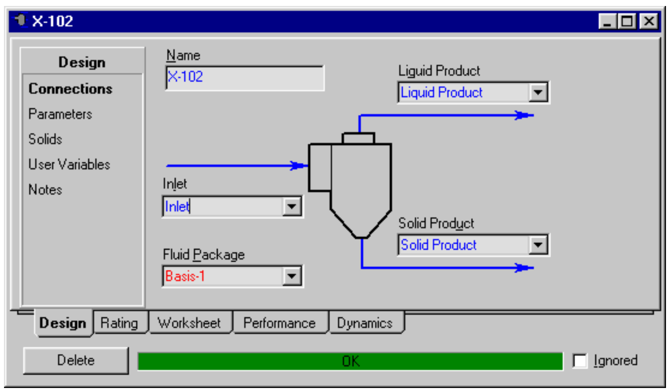


Hydrocyclone icon

3. Double-click the **Hydrocyclone** icon.

The Hydrocyclone property view appears.

Figure 11.16



11.3.2 Design Tab

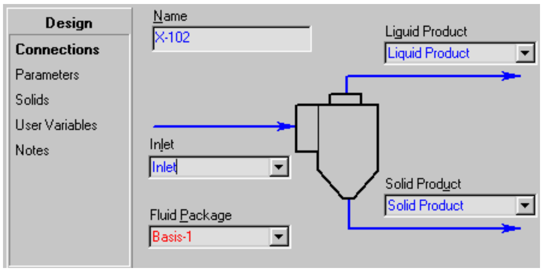
The Design tab contains the following pages:

- Connections
- Parameters
- Solids
- User Variables
- Notes

Connections Page

On the Connections page, you can specify the name of the operation, as well as the feed, liquid product, and solid product streams.

Figure 11.17

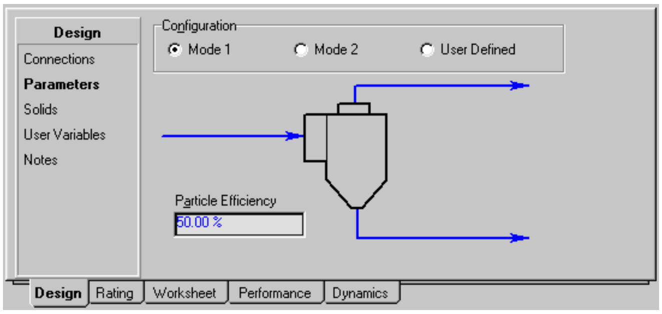


Parameters Page

On the Parameters page, you can specify the following parameters:

Parameter	Description
Configuration	Select either Mode 1, Mode 2 or User Defined.
Particle Efficiency	The percent recovery of the specified solid in the bottoms stream.

Figure 11.18



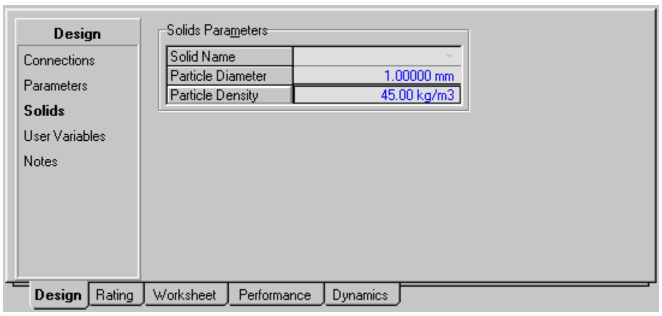
The diameter provided, either from the selected component or from the particle characteristics, is used in the efficiency calculations. For example, if you select an 85% efficiency, 85% of the solids of the specified diameter are recovered. All other solids in the inlet stream are removed at an efficiency related to these parameters.

Solids Page

On the Solids page, the following solids information can be specified:

Parameter	Description
Solid Name	You must provide either the name of a solid already installed in the case, or provide a particle diameter and density.
Particle Diameter and Particle Density	If you do not specify a Solid Name, provide the particle diameter and density.

Figure 11.19



User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

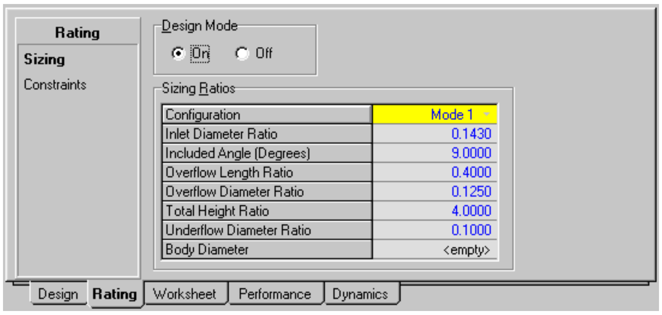
11.3.3 Rating tab

The Rating tab contains the following pages:

- Sizing
- Constraints

Sizing Page

Figure 11.20



The Sizing page contains two groups:

- **Design Mode**. Contains two radio buttons: On and Off.
  The radio buttons enable you to toggle between turning on or turning off the Design Mode option.
- **Sizing Ratio**. Contains a table.
  The table below describes the parameters available on the page:

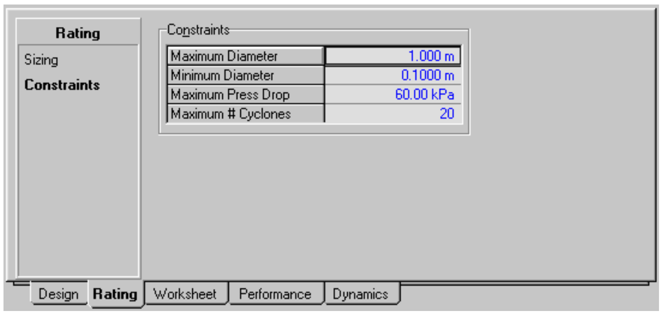
Parameter	Description
Configuration	Select Mode 1, Mode 2 or User Defined. This is also defined on the Parameters page.
Inlet Diameter Ratio	The ratio of the inlet diameter to the body diameter.
Included Angle (Degrees)	The angle of the cyclone slope to the vertical.
Overflow Length Ratio	The ratio of the overflow length to the body diameter.
Overflow Diameter Ratio	The ratio of the overflow diameter to the body diameter.
Total Height Ratio	The ratio of the total height of the apparatus to the body diameter.

Parameter	Description
Underflow Diameter Ratio	The ratio of the underflow diameter to the body diameter.
Body Diameter	If Design Mode is on, then this is automatically calculated, within the provided constraints. If Design Mode is off, then you can specify this value.

Constraints Page

You can specify the minimum and maximum diameter for the Cyclone, applicable only when the On radio button is selected in the Design Mode group.

Figure 11.21



The Maximum Pressure Drop and Maximum Number of Cyclones can also be set on this page.

11.3.4 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

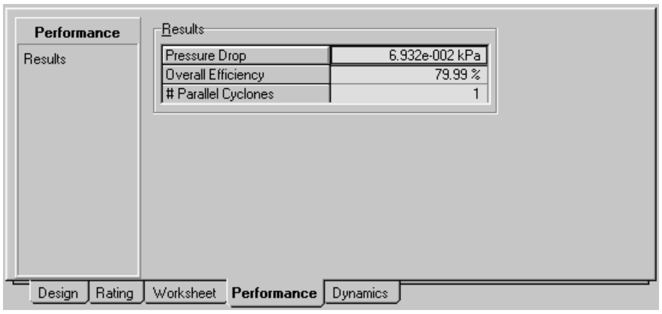
11.3.5 Performance Tab

The Performance tab consists of the Results page.

Results Page

The calculated pressure drop, overall efficiency, and the number of parallel cyclones appear on this page.

Figure 11.22



11.3.6 Dynamics Tab

This unit operation is currently not available for dynamic simulation.

11.4 Rotary Vacuum Filter

The Rotary Vacuum Filter assumes that there is 100% removal of the solid from the solvent stream. This operation determines the retention of solvent in the particle cake, based on the particle diameter and sphericity of your defined solid(s). The diameter and sphericity determines the capillary space in the cake and thus the solvent retention. The solids being separated must be previously specified and installed as components in the stream attached to this operation.

11.4.1 Rotary Vacuum Filter Property View

There are two ways that you can add a Rotary Vacuum Filter to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

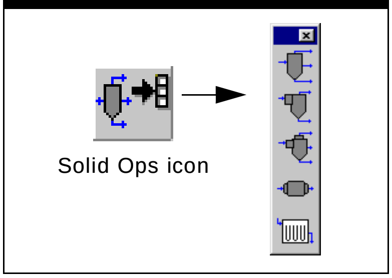
   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Solids Handling** radio button.
3. From the list of available unit operations, select **Rotary Vacuum Filter**.
4. Click the **Add** button.

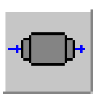
OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.
2. Click on the **Solid Ops** icon. The Solid Operations Palette appears.

Figure 11.23



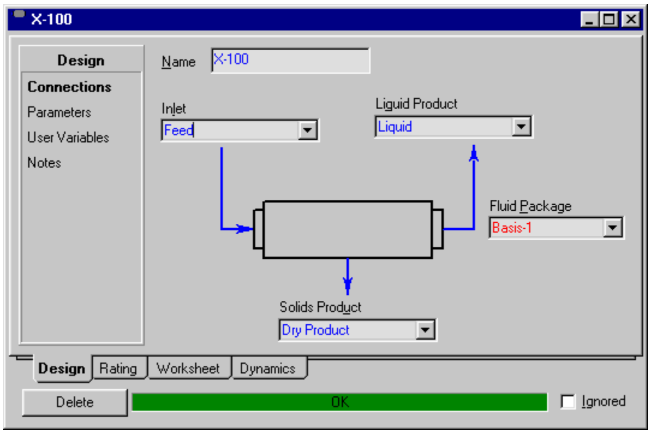


Rotary Vacuum Filter icon

3. Double-click the **Rotary Vacuum Filter** icon.

The Rotary Vacuum Filter property view appears.

Figure 11.24



11.4.2 Design Tab

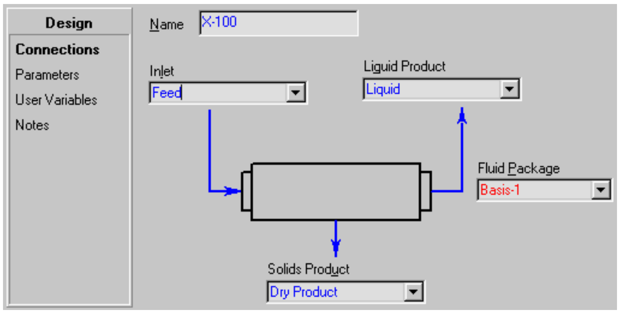
The Design tab contains the following pages:

- Connections
- Parameters
- User Variables
- Notes

Connections Page

On the Connections page, you can define the operation name, as well as the feed, liquid product, and solids product streams.

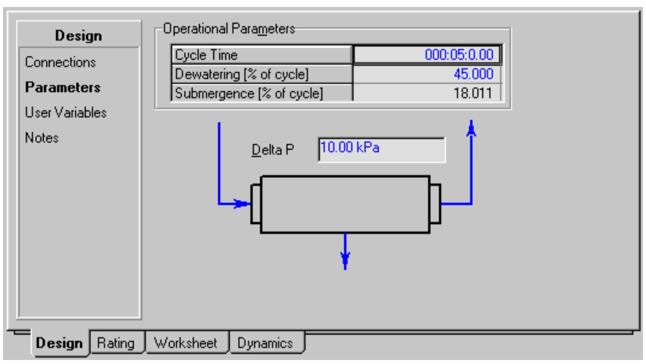
Figure 11.25



Parameters Page

The Rotary Vacuum Filter parameters are described in the table below:

Figure 11.26



Parameter	Description
Cycle Time	The complete time for a cycle (one complete revolution of the cylinder).
Dewatering	The portion of the cycle between the time the cake comes out of the liquid to the time it is scraped, expressed as a percentage of the overall cycle time.

Parameter	Description
Submergence	The percentage of the overall cycle for which the cake is submerged.
Pressure Drop	Pressure drop across the filter.

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

11.4.3 Rating tab

The Rating tab contains the following pages:

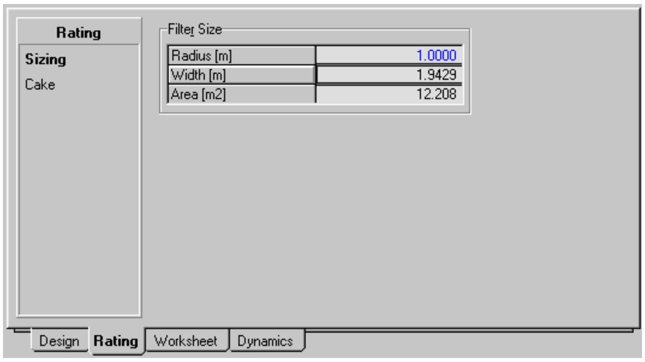
- Sizing
- Cake

Sizing Page

You can specify the following filter size parameters:

Parameter	Description
Filter Radius	The radius of the filter. This defines the circumference of the drum.
Filter Width	The horizontal filter dimension.
Filter Area	The area of the filter.

Figure 11.27

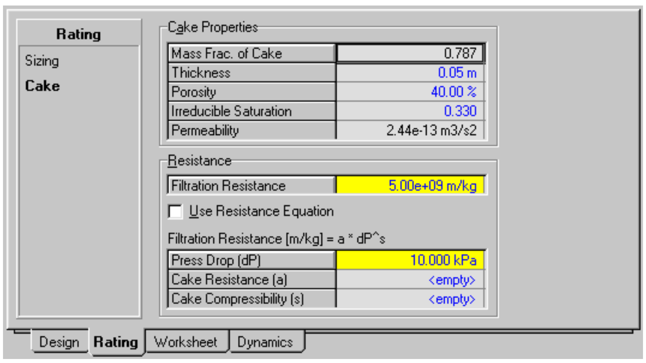


Cake Page

The Cake page consists of two groups:

- Cake Properties
- Resistance

Figure 11.28



You can define the cake properties in the Cake Properties group.

Property	Description
Mass Fraction of Cake	The final solid mass fraction.
Thickness	The thickness of the cake.
Porosity	The overall void space in the cake.

Property	Description
Irreducible Saturation	The solvent retention at infinite pressure drop.
Permeability	If you do not provide a value, HYSYS calculates this from the sphericity and diameter of the solid.

You can define the resistance or use a resistance equation in the Resistance group. Selecting the **Use Resistance Equation** checkbox, which allows HYSYS to calculate the resistance value based on the Filtration Resistance equation.

The Filtration Resistance equation is as follows:

$$Resistance = a(dP)^s \tag{11.1}$$

where:

a, s = *constants*

dP = *pressure drop*

11.4.4 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

11.4.5 Dynamics Tab

This unit operation is currently not available for dynamic simulation.

11.5 Simple Solid Separator

The Simple Solid Separator (Simple Filter) performs a non-equilibrium separation of a stream containing solids. This operation does not perform an energy balance, as the separation is based on your specified carry over of solids in the vapour and liquid streams, and liquid content in the solid product. It should be used when you have an existing operation with known carry over or entrainment in the product streams. The solids being separated must be previously specified and installed as components in the stream attached to this operation.

11.5.1 Simple Solid Separator Property View

There are two ways that you can add a Simple Solid Separator to your simulation:

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Solids Handling** radio button.
3. From the list of available unit operations, select **Simple Solid Separator**.
4. Click the **Add** button.

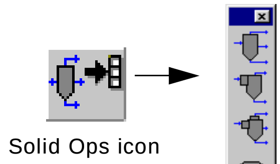
OR

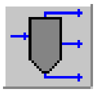
1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing **F4**.

2. Click on the **Solid Ops** icon. The Solid Operations Palette appears.

Figure 11.29



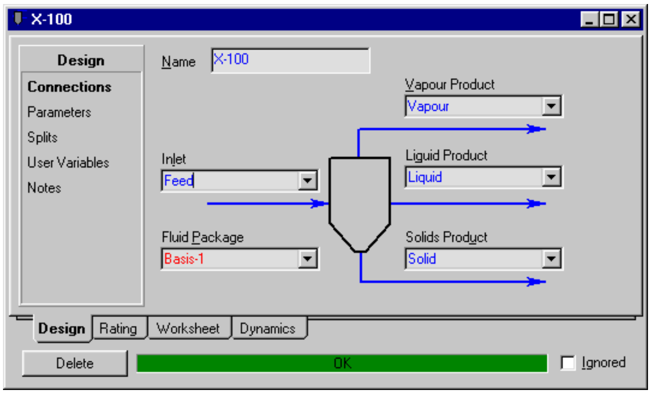


Simple Solid Separator icon

3. Double-click the **Simple Solid Separator** icon.

The Simple Solid Separator property view appears.

Figure 11.30



11.5.2 Design Tab

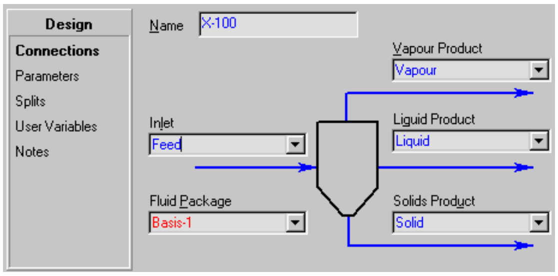
The Design tab contains the following pages:

- Connections
- Parameters
- Splits
- User Variables
- Notes

Connections Page

You can specify the name of the operation, feed stream, and product streams (Vapour, Liquid, Solids) on the Connections page.

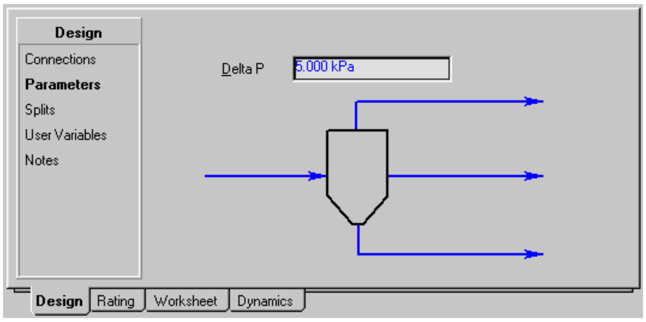
Figure 11.31



Parameters Page

You can specify the pressure drop on the Parameters page.

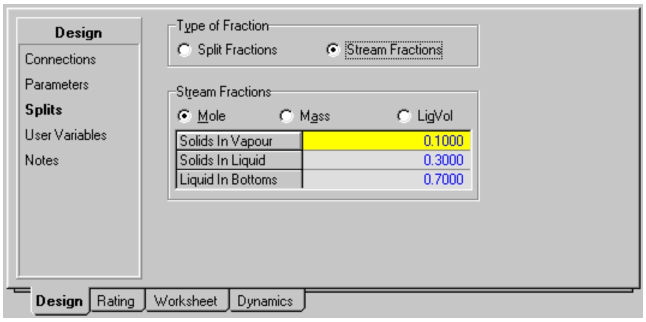
Figure 11.32



Splits Page

On the Splits page, you must choose the split method by defining a Type of Fraction.

Figure 11.33



The types of fraction are described in the table below.

Object	Definition
Split Fractions	Specify the fractional distribution of solids from the feed into the vapour and liquid product streams. The solids fraction in the bottoms are calculated by HYSYS. You must also specify the fraction of liquid in the bottoms (solid product).
Stream Fractions	Enter the mole, mass or liquid volume fraction specification for each of the following: • Total vapour product solids fraction on the specified basis. • Total liquid product solids fraction on the specified basis. • Liquid phase fraction in the bottom product.

In the flowsheet, the streams are not reported as single phase, due to the solid content in the vapour and liquid streams, and the liquid content in the solid product stream.

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for the current operation.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page/Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the specific unit operation, or your simulation case in general.

11.5.3 Rating Tab

This unit operation currently does not have rating features.

11.5.4 Worksheet Tab

Refer to **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

11.5.5 Dynamics Tab

This unit operation is currently not available for dynamic simulation.

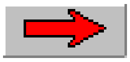
12 Streams

12.1 Energy Stream Property View

Energy streams are used to simulate the energy travelling in and out of the simulation boundaries and passing between unit operations.

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing the **F4** hot key.

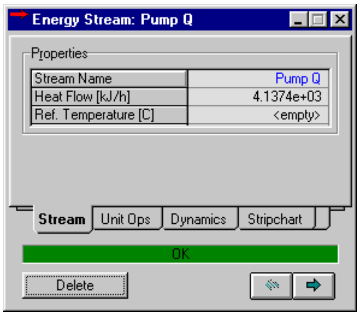


Energy Stream icon

2. Double-click the **Energy Stream** icon.

The Energy Stream property view appears.

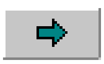
Figure 12.1



The Energy Stream property view contains the following tabs that allow you to define stream parameters, view objects to which the stream is attached, and specify dynamic information:

- Streams
- Unit Ops
- Dynamics
- Stripchart
- User Variables

View Upstream Operation icon



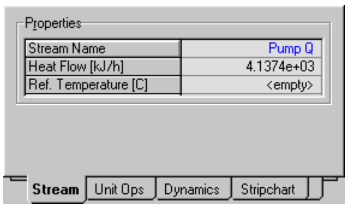
View Downstream Operation icon

As with the material streams, the energy stream property view has the **View Upstream Operation** icon and the **View Downstream Operation** icon that allow you to view the unit operation to which the stream is connected. Energy streams differ from material streams in that if there is no upstream or downstream connection on the stream (which is often the case for the energy stream) the associated icon is not active.

12.1.1 Stream Tab

The Stream tab allows you to specify the Stream Name and Heat Flow for the stream. The figure below shows the Stream tab of the Energy Stream property view.

Figure 12.2

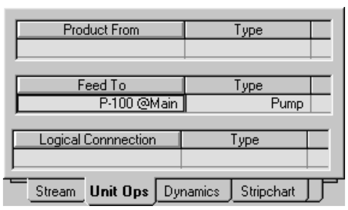


When converting an energy stream to a material stream, all material stream properties are unspecified, except for the stream name.

12.1.2 Unit Ops Tab

The Unit Ops tab displays the Names and Types of all objects to which the energy stream is attached.

Figure 12.3



Both unit operations and logicals are listed. The Unit Ops tab either shows a unit operation in the Product From cell or in the

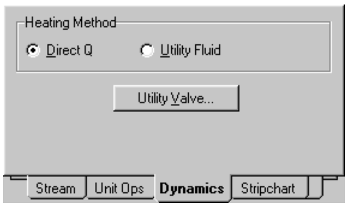
Feed To cell, depending on whether the energy stream receives or provides energy respectively.

You can double-click on either the Product From or Feed To cell to access the property view of the operation attached to the stream.

12.1.3 Dynamics Tab

The options on the Dynamics tab allow you to set the dynamic specifications for a simulation in dynamic mode.

Figure 12.4



In dynamic mode, two different heating methods can be chosen for an energy stream. When the Direct Q radio button is selected, you can specify a duty value. When the Utility Fluid radio button is selected, the duty is calculated from specified properties of a utility fluid.

For a detailed description of the Direct Q and Utility Fluid Heating methods, refer to **Section 7.6 - Valve**.

The Utility Valve button opens the Flow Control Valve (FCV) view for the energy stream.

12.1.4 Stripchart Tab

The Stripchart tab currently does not have any functions.

12.1.5 User Variables Tab

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables tab enables you to create and implement your own user variables for all energy streams.

12.2 Material Stream Property View

Material streams are used to simulate the material travelling in and out of the simulation boundaries and passing between unit operations. For the material stream you must define their properties and composition so HYSYS can solve the stream.

There are three methods to add a Material stream:

1. Select **Flowsheet | Add Stream** command from the menu bar.

OR

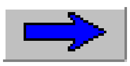
1. Pres the **F11** hot key.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

   You can also open the Object Palette by pressing the **F4** hot key.

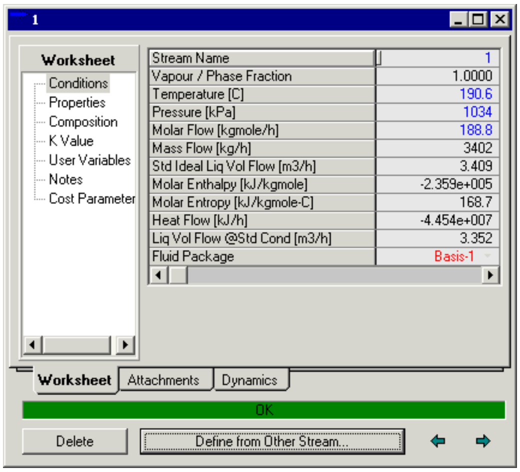
2. Double-click the **Material Stream** icon.



Material Stream icon

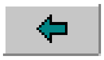
The Material Stream property view appears.

Figure 12.5

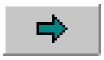


The Material Stream property view contains three tabs and associated pages that allow you to define parameters, view properties, add utilities, and specify dynamic information.

If you want to copy properties or compositions from existing streams within the flowsheet, click the **Define from Other Stream** button. The **Spec Stream As Property View** appears, which allows you to select the stream properties and/or compositions you want to copy to your stream.



View Upstream Operation icon



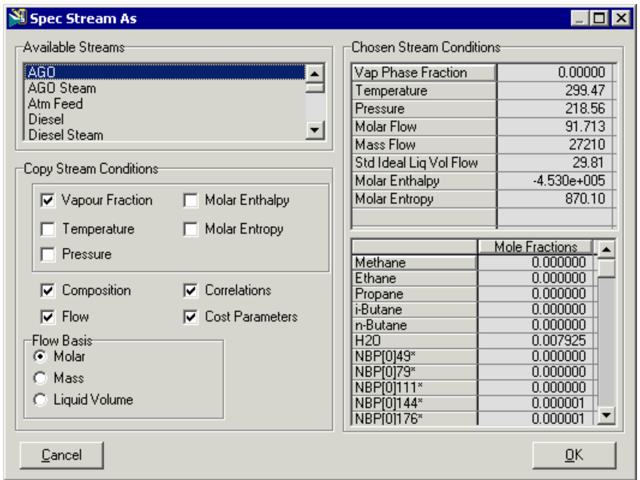
View Downstream Operation icon

The left green arrow is the **View Upstream Operation** icon, which indicates the upstream position. The right green arrow is the **View Downstream Operation** icon, which indicates the downstream position. If the stream you want is attached to an operation, clicking these icons opens the property view of the nearest upstream or downstream operation. If the stream is not connected to an operation at the upstream or downstream end, then these icons open a Feeder Block or a Product Block.

Spec Stream As Property View

The Spec Stream As property view enables you to select properties/information from other streams and use that information to define a stream.

Figure 12.6



- The Available Streams list enables you to select the stream containing the properties you want to copy. You can only select one stream.
- In defining the basic stream properties, you can only select the checkboxes of two of the following stream properties for the new stream: vapour fraction, temperature, pressure, molar enthalpy, or molar entropy.
- The **Composition** checkbox enables you to copy the composition fraction values of the selected stream into the new stream.
- The **Correlations** checkbox enables you to copy the selected stream's correlation configuration into the new stream.
- The **Flow** checkbox enables you to copy the selected stream's flow rate value into the new stream. You can select the flow rate basis you want to copy into the new stream by selecting the appropriate radio button on the Flow Basis group.
- The **Cost Parameters** checkbox enables you to copy the cost parameters values of the selected stream into the new stream.

- The **Cancel** button enables you to exit the property view without accepting any changes.
- The **OK** button enables you to exit the property view and accepts any changes made.

12.2.1 Worksheet Tab

The Worksheet tab has pages that display information relating to the stream properties:

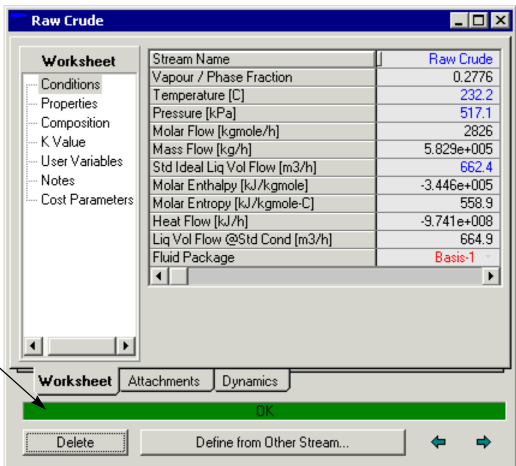
- Conditions
- Properties
- Compositions
- K Value
- Electrolytes

The Electrolytes page is only available if the stream is in an electrolyte system.

- User Variables
- Notes

The figure below shows the Worksheet tab of a solved material stream within a simulation.

Figure 12.7



The green status bar containing **OK** indicates a completely solved stream.

Conditions Page

The Conditions page displays all of the default stream information as it is shown on the Material Streams tab of the Workbook property view. The names and current values for the following parameters appear below:

- Stream Name
- Vapour/Phase Fraction
- Temperature
- Pressure
- Molar Flow
- Mass Flow
- Std Ideal LiqVol Flow
- Molar Enthalpy
- Molar Entropy
- Heat Flow
- LiqVol Flow @ Std Cond
- Fluid Package

In the electrolyte system, the Conditions page contains an extra column. This column displays the property parameters of the stream after electrolyte flash calculations.

HYSYS uses degrees of freedom in combination with built-in intelligence to automatically perform flash calculations. In order for a stream to “flash”, the following information must be specified, either from your specifications or as a result of other flowsheet calculations:

- Stream Composition

Two of the following properties must also be specified; at least one of the specifications must be temperature or pressure:

- Temperature

At least one of the temperature or pressure properties must be specified for the material stream to solve.

- Pressure
- Vapour Fraction

- Entropy

In the electrolyte system, the entropy (S) is always a calculated property.

- Enthalpy

If you specify a vapour fraction of 0 or 1, the stream is assumed to be at the bubble point or dew point, respectively. You can also specify vapour fractions between 0 and 1.

Depending on which of the state variables are known, HYSYS automatically performs the correct flash calculation.

Once a stream has flashed, all other properties about the stream are calculated as well. You can examine these properties through the additional pages of the property view. A flowrate is required to calculate the Heat Flow.

For the reactions involved in the flash and the model used for the flash calculation, refer to **Section 1.6.9 - Electrolyte Stream Flash** in the **HYSYS OLI Interface Reference Guide**.

For an electrolyte material stream, HYSYS conducts a simultaneous phase and reaction equilibrium flash on the stream.

The stream parameters can be specified on the Conditions page or in the Workbook property view. Changes in one area are reflected throughout the flowsheet.

While the Workbook displays the bulk conditions of the stream, the Conditions page, Properties page, and Compositions page also show the values for the individual phase conditions. HYSYS can display up to five different phases.

- **Overall**
- **Vapour**
- **Liquid**. If there is only one hydrocarbon liquid phase, that phase is referred to as liquid.
- **Liquid 1**. This phase refers to the lighter liquid phase.
- **Liquid 2**. This phase refers to the heavier liquid phase.

In HYSYS, the liquid phase, and aqueous phase are internally recognized as Liquid 1, and Liquid 2, respectively. Liquid 1 refers to the lighter phase whereas the heavier phase is recognized as Liquid 2.

- **Aqueous**. In the absence of an aqueous phase, the heavier hydrocarbon liquid is treated as aqueous. When there is only one aqueous phase, that phase is labelled as aqueous.

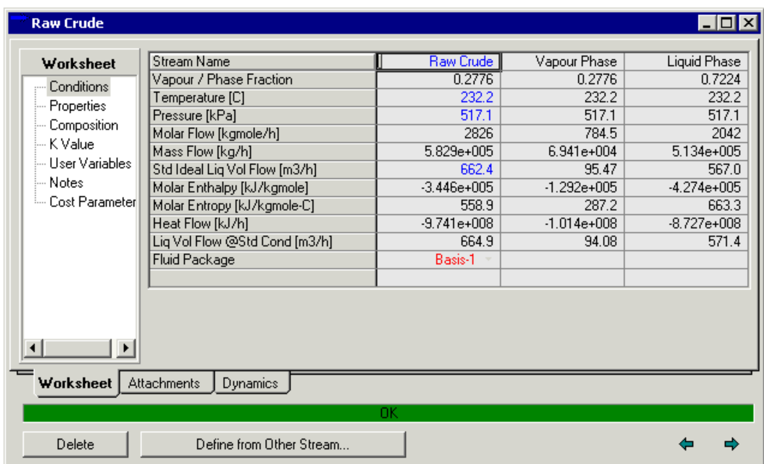
If there is only one hydrocarbon liquid phase, that phase is referred to as liquid.

- **Mixed Liquid**. This phase combines the Liquid phases of all components in a specified stream, and calculates all liquid phase properties for the resulting fluid.

The Mixed Liquid phase does not add its composition or molar flow to the stream it is derived from. This phase is only another representation of existing liquid components.

For example, if you expand the width of the default material stream property view (as shown in the figure below), you can view the hidden phase properties.

Figure 12.8



In this case, the vapour phase and liquid phase properties appear beside the overall stream properties. If there were another liquid phase, it would appear as well.

Rather than expanding the property view, you can use the horizontal scroll bar to view the hidden phase properties.

When you are viewing a stream property view in the column subflowsheet, there is an additional Create Column Stream Spec button on the Conditions page.

For more information on the functionality of the Create Column Stream Spec button, refer to **Section 2.6 - Column Stream Specifications**.

Dynamic Mode

In Dynamic mode, the Manipulate Conditions button appears on the Conditions page of the Material Stream property view.

The Manipulate Conditions button allows you to change the values in a stream if you want to provide a different set of values for when the integrator is started. Normally, you would not have to use this feature. The Manipulate Conditions button is an advanced troubleshooting feature that you can use when you encounter problems, and you want to change the stream values temporarily to affect a downstream operation. You can use this feature, for example, if you ran the simulation and you got really cold temperatures out of a heat exchanger that is causing problems downstream.

This feature can be used on streams that feed into the flowsheet (sits on the boundary) and those that connect operations together. If the stream being changed flows out of a unit operation, its contents are likely overwritten by the upstream operation as soon as you start the integrator.

If the downstream operation is new or had problems solving, changing its feed stream may allow HYSYS to solve the downstream operation or initialize and solve a replacement unit operation.

If you click the **Manipulate Conditions** button, the stream values in the table appear in red. You can then enter in a new temperature (even if the stream had no specifications before). The **Manipulate Conditions** button is also replaced by the **Accept Stream Conditions** button.

If you click the **Accept Stream Conditions** button, HYSYS performs flash calculations again with the initial values you provided. The stream values in the table appear in black as before.

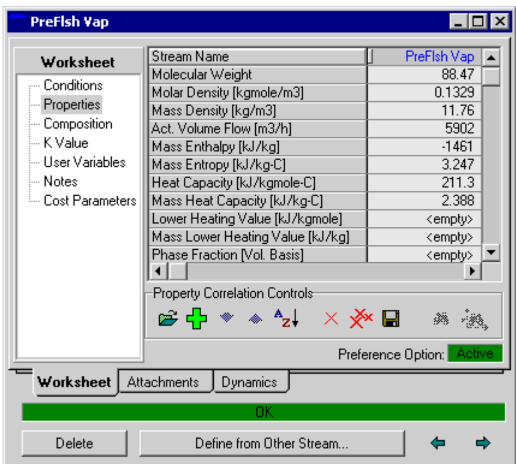
Properties Page

The Properties page displays the properties for each stream phase. The options in the Property Correlation Controls group enables you to manipulate the property correlations displayed on this page for an individual stream. By default the properties from the Conditions page are not available on this page.

Refer to **Section 11.18 - Correlation Manager** in the **HYSYS User Guide** for more information.

You can also manipulate the property correlations displayed in the Properties page using the Correlation Manager.

Figure 12.9



The Properties page contains a table, a Preference Option display field, and a group of icons.

- The table displays the property correlations you select for the stream.
- The **Preference Option** display field is **Active** if the **Activate Property Correlations** checkbox is selected.
  This checkbox can be found on the Options page, Simulation tab of the Session Preferences property view.

- The Property Correlation Controls group contains ten icons. These icons are used to manipulate the property correlations displayed in the table.

You can modify and over-write any existing correlation set using the stream's Property Correlation Controls.

Refer to **Displaying a Correlation Set** section for more information.

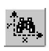
Refer to **Adding a Property Correlation** section for more information.

Refer to **Removing a Property Correlation from the table** section for more information.

Refer to **Creating a Correlation Set** section for more information.

Refer to **Viewing a Property Correlation** section for more information.

Refer to **Viewing All Correlation Plots** section for more information.

Name	Icon	Description
View Correlation Set List		Allows you to select a correlation set.
Append New Correlation		Allows you to add a property correlation to the end of the table.
Move Selected Correlation Down		Allows you to move the selected property correlation one row down the table.
Move Selected Correlation Up		Allows you to move the selected property correlation one row up the table.
Sort Ascending		Allows you to sort the property correlations in the table by ascending alphabetic order.
Remove Selected Correlation		Allows you to remove the selected property correlation from the table.
Remove All Correlations		Allows you to remove all the property correlations from the table.
Save Correlation Set to File		Allows you to save a set of property correlations.
View Selected Correlation		Allows you to view the parameters and status of the selected property correlation.
View All Correlation Plots		Allows you to view all correlation plots for the selected stream.

Adding a Property Correlation

Append New Correlation icon

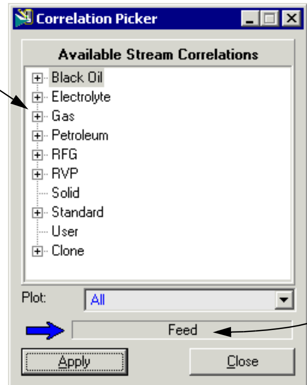
To add a property correlation to the table:

1. Click the **Append New Correlation** icon.

   The Correlation Picker property view appears.

Figure 12.10

HYSYS property correlations have been grouped into categories which target the specific reporting needs of the various process industries.

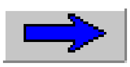


This display field shows the name of the stream the property correlation will be appended to.

2. Select a property correlation that you want to view from the branch list. Click the **Plus** icon  to expand the available correlations list.
3. Click the **Apply** button to append the selected property correlation to the stream.

   If the selected correlation cannot be calculated by that stream's fluid, a message will be sent to the trace window informing the user that this property correlation cannot be added to the stream.
4. Repeat steps #2 to #3 to add another property correlation.
5. When you have completed appending property correlations to the stream, click the **Close** button to return to the stream property view.

To select a different stream to append the property correlations to:



Select Material Stream to Append icon

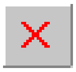
1. Click the **Select Material Stream to Append** icon. The Select Material Stream property view appears.
2. Select the appropriate stream from the object list.

3. Click the **OK** button to return to the Correlation Picker property view. You can now add a property correlation to the selected stream.

   The new selected stream's name also appears in the display field located beside the **Select Material Stream to Append** icon.

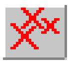
Removing a Property Correlation from the table

To remove property correlations from the table:



Remove Selected Correlation icon

1. Select the property correlation you want to remove in the table.
2. Click the **Remove Selected Correlation** icon. HYSYS removes the selected property correlation from the table.



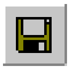
Remove All Correlations icon

You can remove all property correlations in the table by clicking the **Remove All Correlations** icon.

Creating a Correlation Set

To save the property correlations in the table as a set:

1. Add all the property correlations you want to the table.

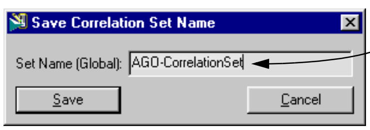


Save Correlation Set to File icon

2. Click the **Save Correlation Set to File** icon. The Save Correlation Set Name property view appears.

   HYSYS automatically enters a name for the property list based on the stream name.

Figure 12.11



For example, the stream called **AGO**, HYSYS automatically enters the default name **AGO-CorrelationSet** for the property list.

3. Enter the name you want for the property list in the **Set Name (Global)** field. Each correlation set name must be unique.

4. Click the **Save** button to save the property list.

> **If you are creating a correlation set for the first time, you are also creating the default file (Support\StreamCorrSets.xml) which will hold all these sets. You can change the name of the file on the Locations page, Files tab of the Session Preferences property view.**

View Correlation Set List icon

The saved correlation set can then be added to other streams using the **View Correlation Set List** icon displayed in each stream property view. You can also add the saved correlation set to all of the streams within the case by using the Correlation Manager.

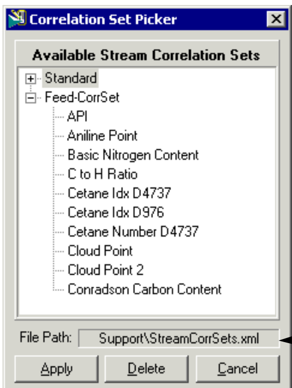
Displaying a Correlation Set

View Correlation Set List icon

To display a correlation set:

1. Click the **View Correlation Set List** icon.
2. The Correlation Set Picker property view appears.

Figure 12.12



The location and name of the file that contains the correlation set is shown in the File Path field. The xml path and file name can be changed using the Session Preferences property view.

Refer to **Section 12.5.2 - Locations Page** from the **HYSYS User Guide**.

> **If the xml file does not exist (you have never created a correlation set before) the window will display "File has not been created." If the xml file does exist but all previous sets have been deleted, the window will display "File is empty."**

3. Select the correlation set you want from the property view.

To see the list of correlations contained within the correlation set, click the associate **Plue** icon ⊞.

4. Click the **Apply** button. The Correlation Set Picker property view will close, and the property correlations contained in the selected correlation set will appear in that streams properties table.

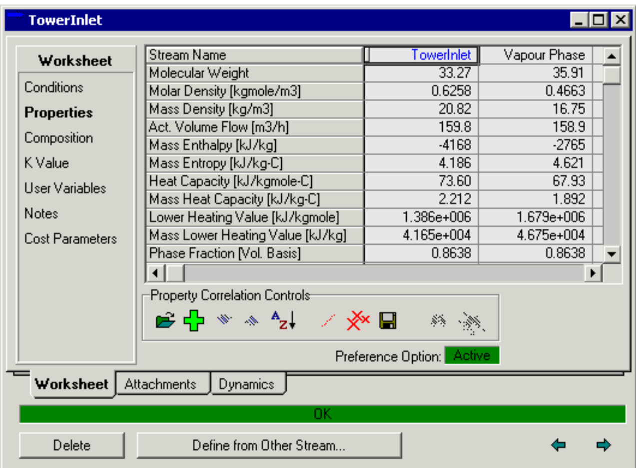
Refer to **Section 1.3 - Object Status & Trace Windows** in the **HYSYS User Guide**, for more information on Trace window.

HYSYS will check each correlation's type against the fluid type of the stream. If a problem occur while appending a correlation from the set, a warning will be sent to the Trace window.

5. Repeat steps #1 to #4 to apply additional correlation sets to your stream.

You can expand the property view or use the scroll bar to view any property correlation phase values, as shown in the figure below.

Figure 12.13



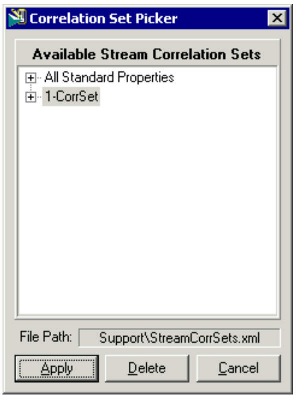
Deleting a Correlation Set

View Correlation Set List icon

To delete a correlation set:

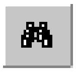
1. Click the **View Correlation Set List** icon.
2. The Correlation Set Picker property view appears.

Figure 12.14



3. Select the correlation set you want to delete.
4. Click the **Delete** button. A window will appear asking you if you are sure you want to delete the set because it cannot be undone.
5. Click the **Yes** button and the Correlation Set Picker property view appears with the chosen set deleted from the list.
6. Click the **Close** icon  to close the Correlation Set Picker property view and return to the stream property view.

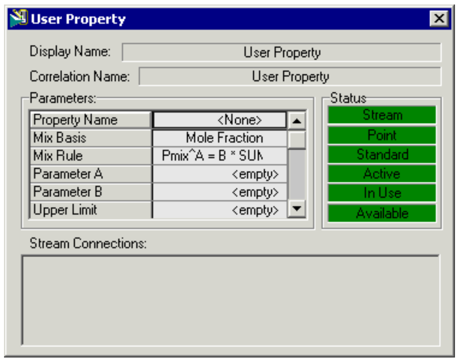
Viewing a Property Correlation



View Selected Correlation icon

When you select a property correlation from the table and click the **View Selected Correlation** icon, the following property view appears.

Figure 12.15



The values shown on the correlation property view cannot be edited. Any configuration parameters available to each property correlation can only be edited using the Correlation Manager property view.

The following table describes the status 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 / Petroleum / RFG / RVP / Solid / Standard / User / Clone	Indicates which correlation type the property correlation resides within in the Available Correlations list.

Status Bar	Description
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.

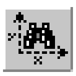
In the Parameters group, you can view the parameters used to calculate the property correlation.

Refer to **Section 11.18 - Correlation Manager** in the **HYSYS User Guide** for more information.

To manipulate the parameter values, you have to access the Correlation Manager property view.

In the Stream Connections group, a list of all the material streams currently using the property correlation is displayed.

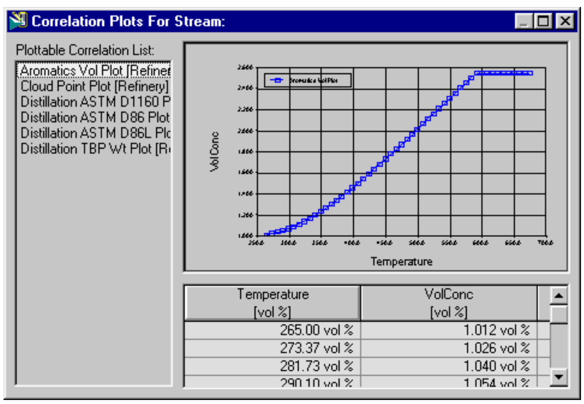
Viewing All Correlation Plots



View All Correlation Plots icon

The **View All Correlation Plots** icon opens the Correlation Plots property view which displays all plottable properties for the stream.

Figure 12.16



Only plottable properties appear on the plots property view, while both point and plot properties appear on the stream properties property view. The plot property view can show only one plot property at a time.

Composition Page

The Composition page enables you to specify and manipulate the stream composition.

> **Blue or red colour text indicates the composition of streams is changeable.**

To specify or change the stream composition do one of the following:

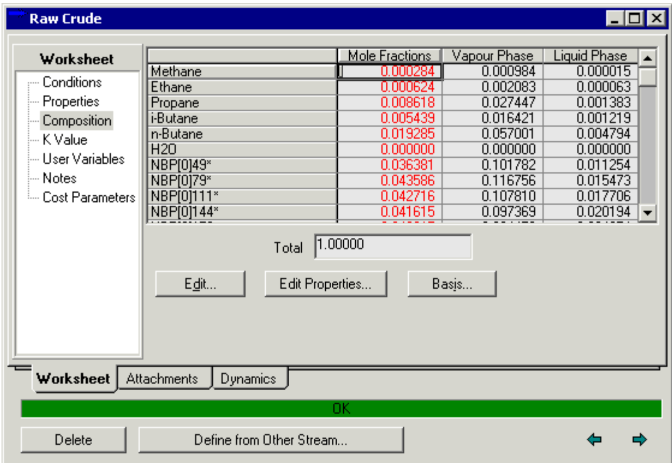
- Click the **Edit** button. The Input Composition property view appears.
- Type a value in a component cell and press **ENTER**. The Input Composition property view appears.

Refer to **Input Composition Property View** section for more information.

> **You cannot edit the compositions for a stream that is calculated by HYSYS. If the composition is calculated by HYSYS, the text colour of the composition value is black and the Edit button will be greyed out.**
>
> **A warning appears if negative mole fraction values occur.**

The figure below shows the mole fractions for each component in the overall phase, vapour phase, and aqueous phase.

Figure 12.17



You can view the composition in a different basis by clicking the Basis button, and selecting the basis you want on the Stream property view.

Stream Property View

The Stream property view contains several radio buttons. The basis type available in HYSYS is represented by each radio button.

Figure 12.18

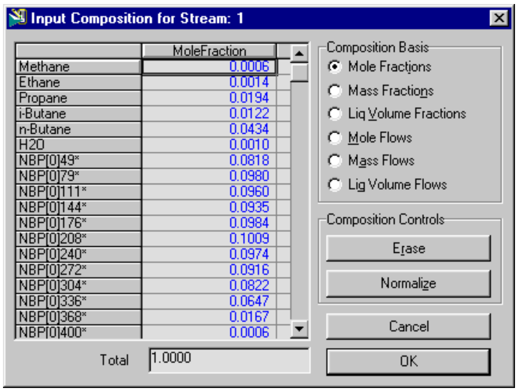
To choose a basis:

1. In the Stream property view, click one of the radio buttons to select a compositional basis.
2. Click the **Close** icon to return to the Composition page. HYSYS displays the stream compositions using the selected basis.

Input Composition Property View

The Input Composition property view allows you to edit the stream compositions.

Figure 12.19



In the Composition Basis group, select the radio button that corresponds to the basis for your stream. In the list of available components, specify the composition of the stream.

The Composition Controls group has two buttons that can be used to manipulate the compositions.

Button	Action
Erase	Clears all compositions.
Normalize	Allows you to enter any value for fractional compositions and have HYSYS normalize the values such that the total equals 1. This button is useful when many components are available, but you want to specify compositions for only a few. When you enter the compositions, click the Normalize button and HYSYS ensures the Total is 1.0, while also specifying any **<empty>** compositions as zero. If compositions are left as **<empty>**, HYSYS cannot perform the flash calculation on the stream. The Normalize button does not apply to flow compositional bases, since there is no restriction on the total flowrate.

The **OK** button closes the Input Composition property view and accepts any specified changes to the stream composition.

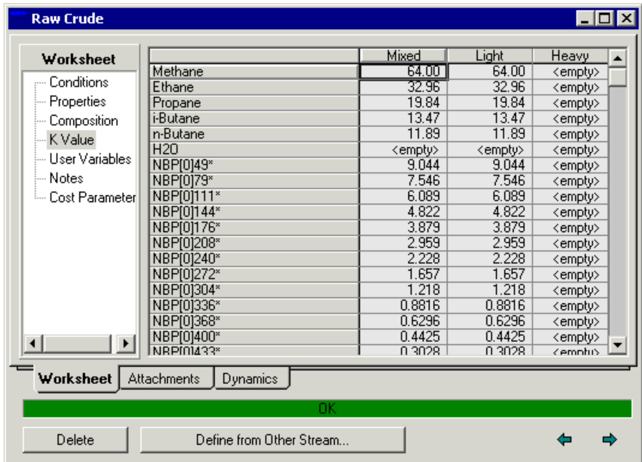
The **Cancel** button closes the property view without accepting any changes.

For fractional bases, clicking the OK button automatically normalizes the composition if all compositions contain a value.

K Value Page

The K Value page displays the K values or distribution coefficients for each component in the stream.

Figure 12.20



A distribution coefficient is a ratio between the mole fraction of component ***i*** in the vapour phase and the mole fraction of component ***i*** in the liquid phase:

$$K_i = \frac{y_i}{x_i} \tag{12.1}$$

where:

K_i = *distribution coefficient*

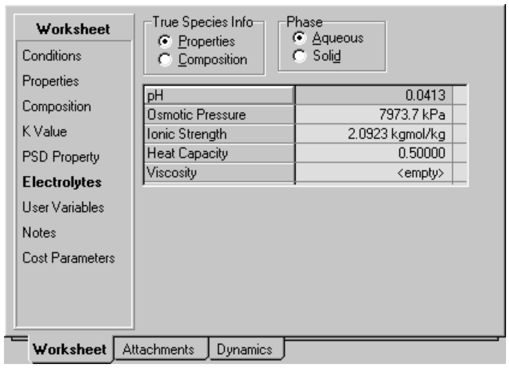
y_i = *mole fraction of component **i** in the vapour phase*

x_i = *mole fraction of component **i** in the liquid phase*

Electrolytes Page

If the stream is associated with an OLI-Electrolyte property package, the Electrolytes page displays electrolytic information.

Figure 12.21



Refer to **Section 1.2.4 - Adding Electrolyte Components** in the **HYSYS Simulation Basis** guide for more information on electrolytes.

The Electrolytes page is available only if the stream is in an electrolyte system.

You can view the electrolyte stream properties or the electrolyte stream composition for aqueous or solid phase. In the True Species Info group, select the appropriate radio button to view the electrolyte stream properties.

The Properties radio button displays the stream fluid phase properties for an electrolyte system. Use the radio buttons in the Phase group to switch between the aqueous phase and the solid phase.

For more information, refer to **Section 1.6 - HYSYS OLI_Electrolyte Property Package** in the **HYSYS OLI Interface Reference Guide**.

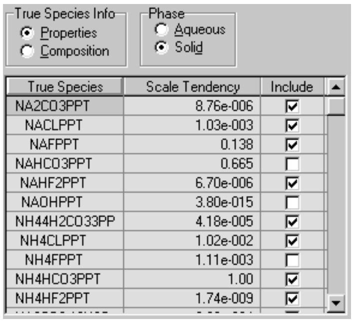
When you click the Aqueous phase radio button, the following aqueous phase related properties appear:

- pH value
- Osmotic Pressure
- Ionic Strength
- Heat Capacity
- Viscosity

To globally include or exclude particular solids in all electrolyte streams, refer to **OLI_Electrolyte Options** section from **Section 2.4.1 - Set Up Tab** in the **HYSYS Simulation Basis** guide.

When you click the Solid phase radio button, you can include or exclude a particular solid in the current stream equilibrium flash calculation.

Figure 12.22



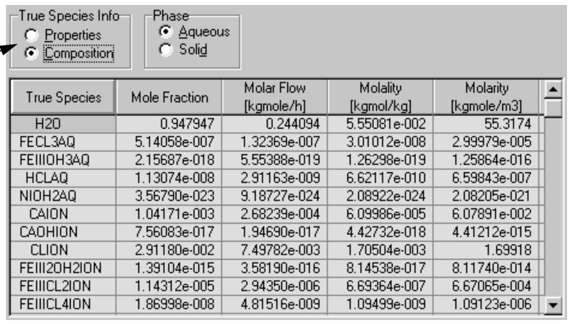
True Species	Scale Tendency	Include
NA2CO3PPT	8.76e-006	☑
NACLPPT	1.03e-003	☑
NAFPPT	0.138	☑
NAHCO3PPT	0.665	☐
NAHF2PPT	6.70e-006	☑
NAOHPPT	3.80e-015	☐
NH44H2CO33PP	4.18e-005	☑
NH4CLPPT	1.02e-002	☑
NH4FPPT	1.11e-003	☐
NH4HCO3PPT	1.00	☑
NH4HF2PPT	1.74e-009	☑

The value of Scale Tendency Index as shown in the table is a measure of the tendency of a solid species forming at the specified conditions. Solid with a scaling tendency greater than one forms if the solid formation is governed by equilibrium (as opposed to kinetics), and if there are no other solids with a common cation or anion portion which also has a scaling tendency greater than one.

The Composition radio button displays the component name, molar fraction, molar flow, or molality and molarity of all the components in the stream for aqueous or solid phase in a table.

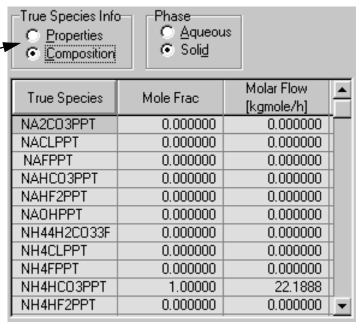
Figure 12.23

If you select the Aqueous radio button, the component list including ionic component(s) appears in the table.



True Species	Mole Fraction	Molar Flow [kgmole/h]	Molality [kgmol/kg]	Molarity [kgmole/m3]
H2O	0.947947	0.244094	5.55081e-002	55.3174
FECL3AQ	5.14058e-007	1.32369e-007	3.01012e-008	2.99979e-005
FEIIIOH3AQ	2.15687e-018	5.55388e-019	1.26298e-019	1.25864e-016
HCLAQ	1.13074e-008	2.91163e-009	6.62117e-010	6.59843e-007
NIOH2AQ	3.56790e-023	9.18727e-024	2.08922e-024	2.08205e-021
CAION	1.04171e-003	2.68239e-004	6.09986e-005	6.07891e-002
CAOHION	7.56083e-017	1.94690e-017	4.42732e-018	4.41212e-015
CLION	2.91180e-002	7.49782e-003	1.70504e-003	1.69918
FEIII2OH2ION	1.39104e-015	3.58190e-016	8.14538e-017	8.11740e-014
FEIIICL2ION	1.14312e-005	2.94350e-006	6.69364e-007	6.67065e-004
FEIIICL4ION	1.86998e-008	4.81516e-009	1.09499e-009	1.09123e-006

If you select the Solid radio button, a component list including precipitate (PPT) and hydrated (-nH_2O) solid appears with only mole fraction and mole flow.



True Species	Mole Frac	Molar Flow [kgmole/h]
NA2CO3PPT	0.000000	0.000000
NACLPPT	0.000000	0.000000
NAFPPT	0.000000	0.000000
NAHCO3PPT	0.000000	0.000000
NAHF2PPT	0.000000	0.000000
NAOHPPT	0.000000	0.000000
NH44H2CO33F	0.000000	0.000000
NH4CLPPT	0.000000	0.000000
NH4FPPT	0.000000	0.000000
NH4HCO3PPT	1.00000	22.1888
NH4HF2PPT	0.000000	0.000000

User Variables Page

For more information refer to **Section 1.3.8 - User Variables Page / Tab**.

The User Variables page enables you to create and implement your own user variables for all material streams.

Notes Page

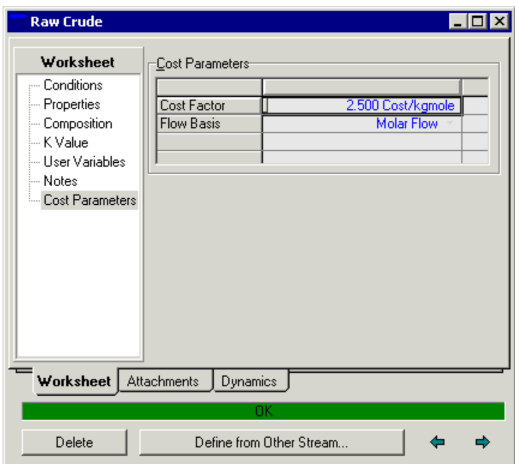
For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor that allows you to record any comments or information regarding the material stream or the simulation case in general.

Cost Parameters Page

You can enter a cost factor value for the stream in the Cost Parameters page. You can also choose the flow basis associated with the cost factor from the Flow Basis drop-down list.

Figure 12.24

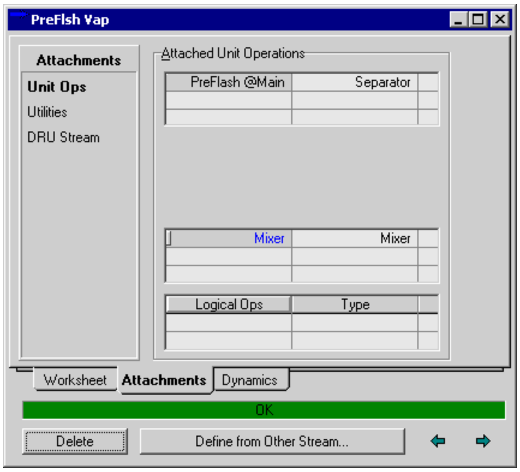


12.2.2 Attachments Tab

Unit Ops Page

The Unit Ops page allows you to view the names and types of unit operations and logicals to which the stream is attached.

Figure 12.25



The property view uses three groups:

- The units from which the stream is a product.
- The units to which the stream is a feed.
- The logicals to which the stream is connected.

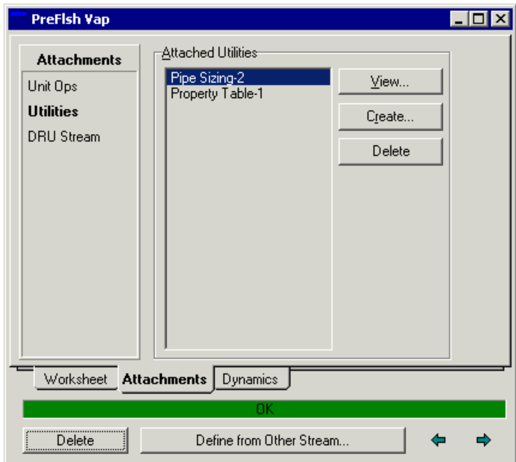
You can access the property view for a specific unit operation or logical by double-clicking in the Name cell or Type cell.

Utilities Page

Refer to **Chapter 14 - Utilities** for more information on the utilities available in HYSYS.

The Utilities page allows you to view and manipulate the utilities attached to the stream.

Figure 12.26



The Utilities page allows you to do the following:

- Attach utilities to the current stream.
- View existing utilities that are attached to the stream.
- Delete existing utilities that are attached to the stream.

Refer to **Section 7.26 - Utilities** in the **HYSYS User Guide** for more information on adding, viewing, and deleting utilities.

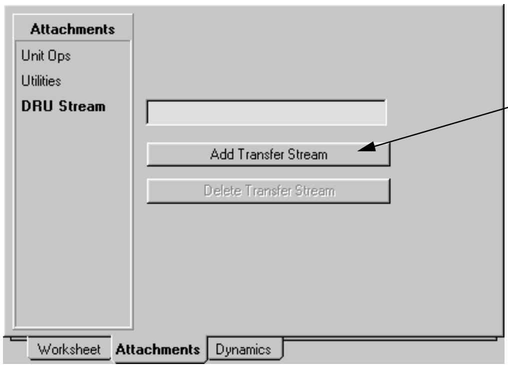
Only the Create button is available all the time.

The View and Delete buttons are greyed out until you select a utility from the list.

DRU Stream Page

The DRU stream facilitates running a given set of unit operations under different stream conditions. The information gathered from the run are stored within the DRU stream, and can be either user input or acquired from the RTO system directly.

Figure 12.27



When the Add Transfer Stream button is enabled, the Delete Transfer Stream button is disabled, and vice-versa.

The DRU stream is applicable for data reconciliation problems.

The DRU stream is used for data reconciliation to hold different states of streams. During data reconciliation, measured data of DCS tags can be obtained under different stream states (for example, temperature or pressure). The DRU stream can also perform flash calculations as other HYSYS streams do.

If you want to use the DRU stream to hold data, the number of data sets needs to be equal to the number of data sets of DCS tags. You can create data sets for streams and set values to stream states.

Each data set behaves as a stream (for example, the data set contains automatic flash calculation and freedom control).

The Data Rec utility controls the updating of its associated streams with the correct data corresponding to the data set being evaluated at that point in time.

The Add Transfer Stream button and Delete Transfer Stream button are solely for On-Line applications.

- Clicking the Add Transfer Stream button creates a DRU Stream (Data Reconciliation Stream) such that you can move the information for the stream as a block between HYSYS cases, or instances of HYSYS.
- Clicking the Delete Transfer Stream button removes the DRU Stream.

12.2.3 Dynamics Tab

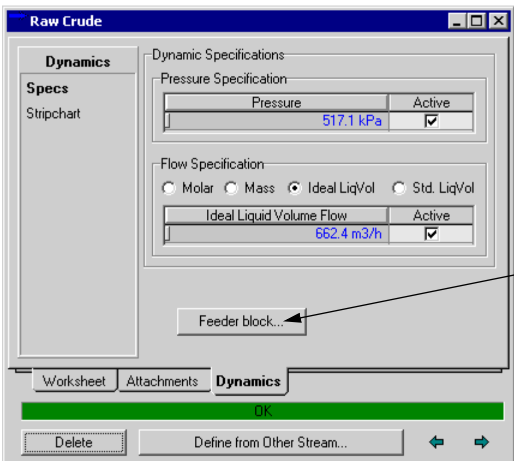
The Dynamics tab displays the pressure and flow specifications for the material stream, and enables you to generate the strip chart for a set of variables.

You must be in dynamic mode for any of these specifications to have an effect on the simulation.

Specs Page

The Specs page allows you to add a pressure and a flow specification for the stream.

Figure 12.28



Feeder block or Product block button appears only when the stream is a flowsheet boundary stream.

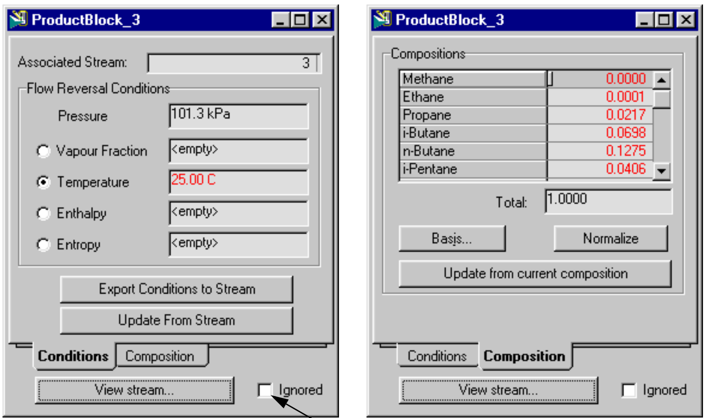
If the **Active** checkbox is selected for a specification, the value of the specification appears in blue and you can modify the value. If the **Active** checkbox is cleared, the value is appears in black and is calculated by HYSYS. Default stream conditions are shown in red.

Feed and Product Blocks

A flowsheet boundary stream is a stream which has only one unit operation attached to it. If a material stream is a flowsheet boundary stream, a Feeder block button or Product block button appears in the Specs page of the Dynamics tab. A flowsheet boundary stream can be the feed or product of the model.

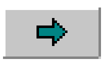
Depending on whether the flowsheet boundary stream is a feed or a product, the **Dynamics** tab contains either a **Feeder block** button or a **Product block** button. The figure below shows a Product Block property view of a material stream.

Figure 12.29



To ignore the Feed or Product block operations, select the **Ignored** checkbox.

View Upstream Operation icon



View Downstream Operation icon

You can also access the Feeder Block property view by clicking the **View Upstream Operation** icon on the stream property view. Similarly, you can also access the Product Block property view by clicking the **View Downstream Operation** icon.

The Product Block property view displays flow reversal conditions of the material stream which you can specify. If simulation conditions are such that the product stream flow becomes negative, HYSYS recalls the stream conditions stored in the Product block and performs a rigorous flash on the product stream to determine the other stream conditions.

When process conditions in the simulation cause the feed flow to reverse, the feed stream conditions are calculated by the downstream operation. The Feeder Block property view is used to restore desired feed conditions and compositions if the feed stream reverses and then becomes feed again.

The Feeder Block and Product Block have similar property views. You can specify the stream conditions as follows:

Required Feed and Product Block Specifications	
Conditions Tab	Specify one of the following: • Temperature • Vapour Fraction • Entropy • Enthalpy
Composition Tab	Specify the stream composition.

Since the pressure of the stream remains the same after the product stream flow reverses, the pressure value does not need to be specified. With this information, the stream is able to perform flash calculations on the other stream properties.

Both the Feeder Block property view and Product Block property view have three buttons that allow you to manipulate the direction of stream conditions between the material stream and the block. The table below briefly describes each button.:

Block Button	Action
Export Conditions to Stream	Copies stream conditions stored in the block to the material stream.
Update From Stream	Copies the current stream conditions from the material stream to the block.
Update from Current Composition	Copies only the stream composition from the material stream to the block.

Stripchart Page

Refer to **Section 1.3.7 - Stripchart Page/Tab** for more information.

The Stripchart page allows you to select and create default strip charts containing various variable associated to the material stream.

13 Subflowsheet Operations

13.1 Introduction

The subflowsheet operation uses the multi-level flowsheet architecture and provides a flexible, intuitive method for building the simulation. Suppose you are simulating a large processing facility with a number of individual process units and instead of installing all process streams and unit operations into a single flowsheet, you can simulate each process unit inside its own compact subflowsheet.

Once a subflowsheet operation is installed in a flowsheet, its property view becomes available just like any other flowsheet object. Think of this property view as the "outside" property view of the "black box" that represents the subflowsheet. Some of the information contained on this property view is the same as that used to construct a Template type of Main flowsheet. Naturally this is due to the fact that once a Template is installed into another flowsheet, it becomes a subflowsheet in that simulation.

Whether the flowsheet is the Main flowsheet of a simulation case, or it is contained in a subflowsheet operation, it possesses the following components:

- **Fluid Package**. An independent fluid package, consisting of a Property Package, Components, and so forth. It is not necessary that every flowsheet in the simulation have its own separate fluid package. More than one flowsheet can share the same fluid package.
- **Flowsheet Objects**. The inter-connected topology of the flowsheet. Unit operations, material and energy streams, utilities, and so forth.
- **A Dedicated PFD**. A HYSYS property view presenting a graphical representation of the flowsheet, showing the inter-connections between flowsheet objects.
- **A Dedicated Workbook**. A HYSYS property view of tabular information describing the various types of flowsheet objects.
- **A Dedicated Desktop**. The PFD and Workbook are home property views for this Desktop, but also included are a menu bar and a toolbar specific to either regular or Column subflowsheets.

13.2 MASSBAL Subflowsheet

HYSYS solves as a sequential modular solver. Unit operations must have specific degrees of freedom in order for the unit operation to solve. MASSBAL is a simultaneous solver. In MASSBAL, a completely specified problem requires that there be no degrees of freedom remaining for the flowsheet, however, the specifications are restricted on a unit by unit basis and can be specified anywhere in the flowsheet.

The task is to allow you to use MASSBAL within a HYSYS interface. The design has two modes of operation:

- **Generating Cases via the MASSBAL flowsheet**. Within the MASSBAL flowsheets in HYSYS, you can create HYSYS unit operations that can either be solved sequentially or simultaneously. You can select unit operations and streams from the Object Palette and create the PFD in the MASSBAL flowsheet. You can also make a list of specifications within the MASSBAL flowsheet. Upon calculating simultaneously, MASSBAL uses the specifications to create results files.
- **Reading in Previously Created Cases**. You also have the option of reading in previously created cases into HYSYS. You can run previously created cases but cannot modify the cases through the HYSYS interface. You have to modify the *.dat files directly.

Other important information:

- **Solving Backwards**. All source streams in the MASSBAL flowsheet have to be fully specified (Phase Rule has to be satisfied for each stream). Specifications can be made elsewhere in the flowsheet.
- **Thermo Interfaces**. MASSBAL has many different possible stream definitions (for example, Chemical, VLE, Fluid, Food, and Pulp). The only one used in HYSYS, however, is the VLE stream type. Thus, in order for MASSBAL to use HYSYS to perform its thermodynamic calculations, callback functions have been set up to deal with flashes and property calculations of individual components and streams.

13.2.1 Adding a MASSBAL Subflowsheet

There are two ways you can add a MASSBAL subflowsheet to your simulation.

1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.

2. Click the **Sub-Flowsheets** radio button.
3. From the list of available unit operations, select **MassBal SubFlowsheet**.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

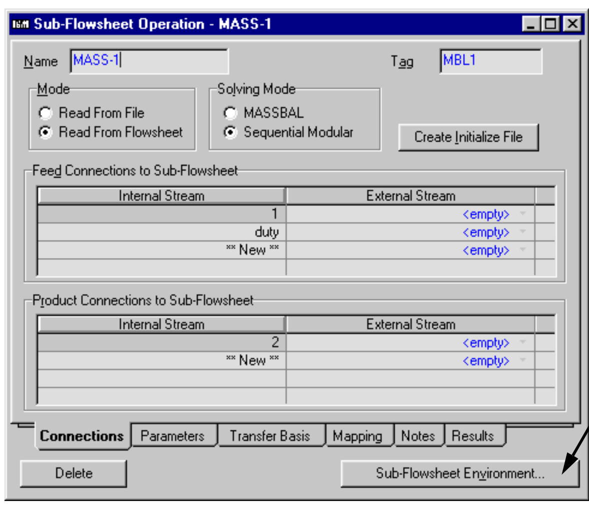
   You can also open the Object Palette by pressing **F4**.

2. Double-click on the **MassBal** icon on the Object Palette.

MassBal icon

The MASSBAL property view appears.

Figure 13.1



You can enter the MASSBAL subflowsheet environment by clicking this button.

The MASSBAL property view consists of the following tabs:

- Connections
- Parameters
- Transfer Basis
- Mapping
- Notes
- Results

13.2.2 Connections Tab

The Connections tab allows you to choose between opening a previously created case or generating a case in the MASSBAL flowsheet.

You can change the name of the MASSBAL operation or the Tag name by typing the new name in the Name field or Tab field respectively.

The table below briefly describes the four groups on the Connections tab.

For more information, refer to **Reading in Previously Created Cases** and **Generating Cases via the MASSBAL Flowsheet** sections.

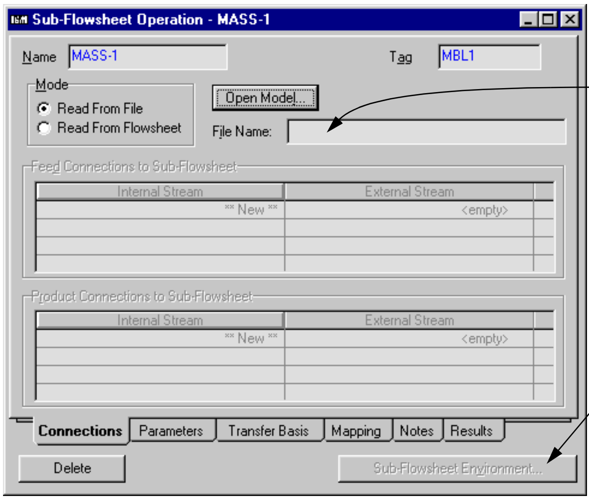
Group	Description
Mode	Click on one of the radio buttons to select the mode you want to use. There are two radio buttons: • **Read from File**. Select this radio button if you want to use an existing case. • **Read from Flowsheet**. Select this radio button if you want to generate a case in the MASSBAL flowsheet.
Solving Mode	Select one of the radio buttons to choose the mode you want to use: • MASSBAL • Sequential Modular The Solving Mode group is only available if you select the **Read from Flowsheet** radio button in the Mode group.
Feed Connections to Sub-Flowsheet	Allows you to select the external stream you want to enter the MASSBAL subflowsheet. In the External Stream column, you can either type in the name of the stream or you can select a pre-defined stream from the drop-down list.
Product Connections to Sub-Flowsheet	Allows you to select the external stream you want to exit the MASSBAL subflowsheet. In the External Stream column, you can either type in the name of the stream or you can select a pre-defined stream from the drop-down list.

Reading in Previously Created Cases

To read a previously created case in the MASSBAL flowsheet, a *.dat file must be provided containing information of a case.

1. On the **Connections** tab of the MASSBAL property view, click the **Read From File** radio button in the Mode group.

Figure 13.2



The File Name display field shows the path and the name of file being used.

The **Sub-Flowsheet Environment** button appears greyed because you are not allowed to modify the case within the HYSYS interface.

2. Click the **Open Model** button. The Choose a MASSBAL File property view appears.
3. From the list of file names, select the *.dat file containing the information you want
4. Click the **OK** button.

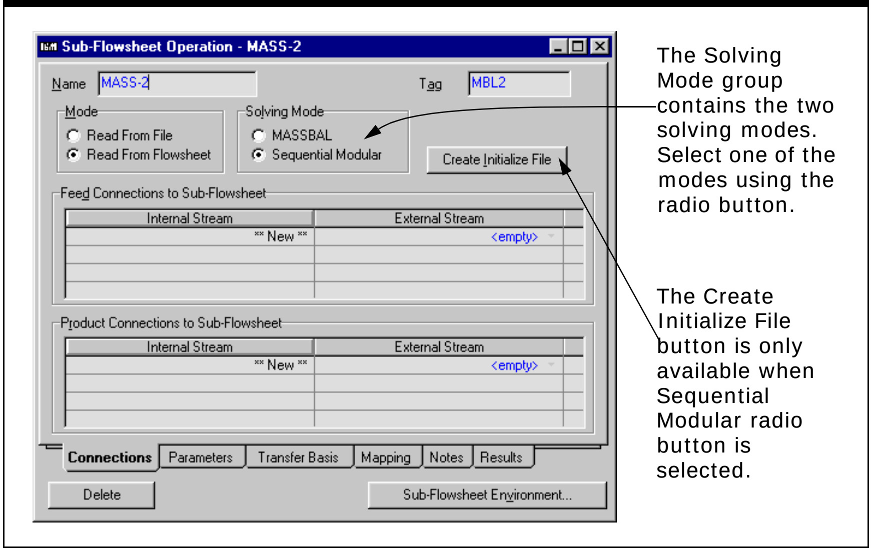
   The name of the file and path appears in the **File Name** field of the MASSBAL property view.

Generating Cases via the MASSBAL Flowsheet

To generate cases via the MASSBAL flowsheet, the MASSBAL flowsheet is like a subflowsheet or template. You can enter the MASSBAL flowsheet environment and build the simulation case just like a flowsheet.

1. On the **Connections** tab of the MASSBAL property view, select the **Read From Flowsheet** radio button in the Mode group.

Figure 13.3



2. In the Solving Mode group, select one of the radio buttons to set how you want to write out to streams:
   - **MASSBAL**. The MASSBAL flowsheet writes out calculated values to streams and pertinent unit operations.

     You can also view the MASSBAL results for the PH1 and PH2 files on the Results tab.
   - **Sequential Modular**. The MASSBAL flowsheet solves using the HYSYS solver. You have the option of running MASSBAL on the existing operations. The only difference is that the stream results of MASSBAL won't be printed to any of the streams in Sequential Modular mode.

     Click the **Create Initialize File** button to create a *.sav file containing initial estimates for the solver calculations.

The *.sav file can be accessed using the **Use Initialize File** checkbox in the **Parameters** tab.

3. Click the **Sub-Flowsheet Environment** button to enter the MASSBAL flowsheet environment.

A MASSBAL Object in HYSYS is a flowsheet object (similar to a template object) that holds all streams or unit operations subject to the equation based solver.

Refer to **Chapter 8 - HYSYS Objects** in the **HYSYS User Guide** for more information regarding installing streams and operations.

4. Enter the material streams and unit operations in the MASSBAL PFD to create the simulation case.

   The stand alone MASSBAL application uses a graphical interface to create a *.dat file. The *.dat file is a text file containing the streams, unit operations, connections, and any specifications the case may have. The *.dat file is used by HYSYS to generate simulation results.

   HYSYS is able to translate the following unit operations: Separator, Heat Exchanger, Valve, Heater, Cooler, Compressor, Expander, Pump, Mixer, Tee, Recycle, Adjust, and Set.

The concept of the stream in HYSYS is different from that in MASSBAL. HYSYS streams are flowsheet objects with properties/characteristics (and can exist without unit operations) whereas MASSBAL streams are connections between unit operations. Special streams known as Sources feed into unit operations and are fully defined for VLE cases. Streams that exit the flowsheet are known as Sinks.

In MASSBAL, convention dictates that streams are defined as either feeds to a unit operation or products of a unit operation. In generating identifiers for streams, HYSYS has associated each stream as the product of the immediate upstream unit operation. This works for all streams except Source streams, which are fully defined.

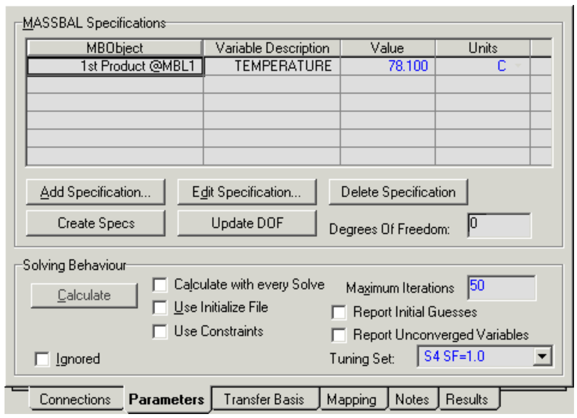
Refer to **Section 13.2.3 - Parameters Tab** for more information.

5. On the **Parameters** tab:
   - Select the option for the convergence process.
   - Enter specifications used for the MASSBAL equation-based solver.
   - Manipulate the solving behaviour of the MASSBAL flowsheet.

13.2.3 Parameters Tab

The Parameters tab allows you to specify variables used for the MASSBAL equation-based solver, select derivative options to help the calculations converge, and manipulate the solving behaviour.

Figure 13.4



MASSBAL Specifications Group

The MASSBAL Specifications group contains a table and buttons that enables you to manipulate the specifications for the MASSBAL equation-based solver.

- The table contains the list of variables that are used for the specifications. The Value column allows you to specify the variables value. The Units column allows you to specify the units for the variable values entered.
- The Add Specification button enables you to add variables.
- The Edit Specification button enables you to change the selected variable in the table.
- The Delete Specification button enables you to remove the selected variable in the table.
- The Update DOF button recalculate and updates the degree of freedom value of the subflowsheet.

- The Degrees of Freedom field displays the number of degrees of freedom available. The number of degrees of freedom is not applicable to the Sequential Modular solving mode.

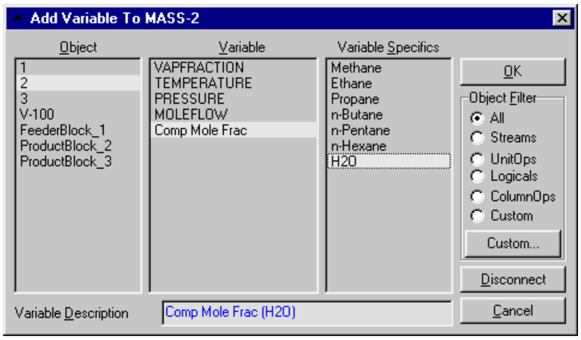
Adding a Specification

To add a MASSBAL specification, do the following:

1. On the **Parameters** tab of the MASSBAL property view, click the **Add Specification** button in the MASSBAL Specifications group.

   The Add Variable To Mass property view appears.

Figure 13.5



Refer to **Section 1.3.9 - Variable Navigator Property View** for more information.

The Add Variable To Mass property view is similar to the Variable Navigator property view.

2. Select the variable you want to specify.
3. Click the **OK** button.

   You are automatically returned to the Parameters tab. The table in the MASSBAL Specifications group displays the variable selected from the Add Variable To Mass property view.

Editing a Specification

To edit a MASSBAL specification:

1. On the **Parameters** tab of the MASSBAL property view, select the variable you want to edit from the table.
2. Click on the **Edit Specification** button in the MASSBAL Specifications group.

   The Add Variable To Mass property view appears.

Refer to **Section 1.3.9 - Variable Navigator Property View** for more information.

> **The Add Variable To Mass property view is similar to the Variable Navigator property view.**

3. Select the new variable you want to specify and click the **OK** button.

   You are automatically returned to the Parameters tab. The table in the MASSBAL Specifications group displays the new variable selected from the Add Variable To Mass property view.

Deleting a Specification

To delete a MASSBAL specification:

1. On the **Parameters** tab of the MASSBAL property view, select the variable you want to delete from the table.
2. Click on the **Delete Specification** button in the MASSBAL Specifications group. The selected variable is removed from the table.

Solving Behaviour Group

The Solving Behaviour group contains options to manipulate the solving behaviour of the MASSBAL flowsheet.

> **Most options (exception Ignored, Report Initial Guesses, Report Unconverged Variables, and Tuning Set) in the Solving Behaviour group are only available if the Read From Flowsheet and MASSBAL radio buttons are selected.**

- **Calculate with every Solve**. When this checkbox is selected, the solver behaves like a regular HYSYS case. If there are changes upstream of the MASSBAL operation, then it automatically resolves. When the checkbox is cleared, you are required to click the **Calculate** button to get MASSBAL to recalculate each time.

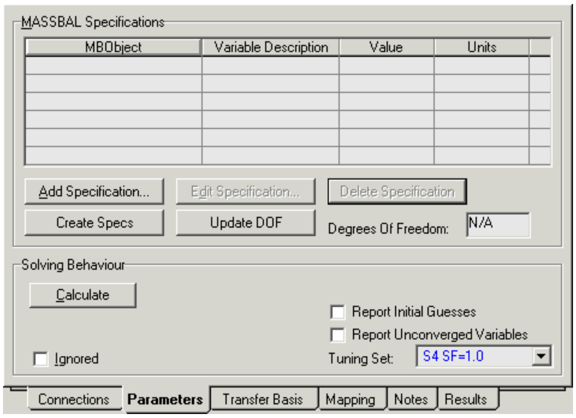
  The **Calculate** button is greyed out and made unavailable when the **Calculated with every Solve** checkbox is selected.
- **Maximum Iterations**. The value in this field sets the maximum number of iterations that the solver is allowed to perform regardless if the solution is converged or not. You can change the value in this field.
- **Use Initialize File**. When this checkbox is selected, a *.sav file is used as initial estimates for the Mass solver. The *.sav file is created when you click the **Create Initial File** button on the Connections tab. Activating this option can aid in the convergence of cases by providing the solver with better initial values.

The Create Initial File button is only available when the solving mode is Sequential Modular.

- **Ignored**. Select this checkbox to ignore the options and settings in the Solving Behaviour group.

If the Read From File radio button is selected, the degree of freedom option is ignored and the solving options in the **Parameter** tab are limited, as shown in the figure below:

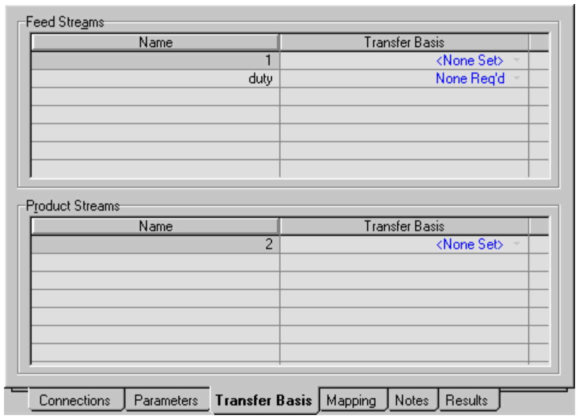
Figure 13.6



13.2.4 Transfer Basis Tab

The transfer basis for each feed and product stream is listed on the Transfer Basis tab.

Figure 13.7



The Transfer Basis is also useful in controlling VF, T, or P calculations in column subflowsheet boundary streams with close boiling or nearly pure compositions.

Refer to **Section 13.3.4 - Transfer Basis Tab** for more Information.

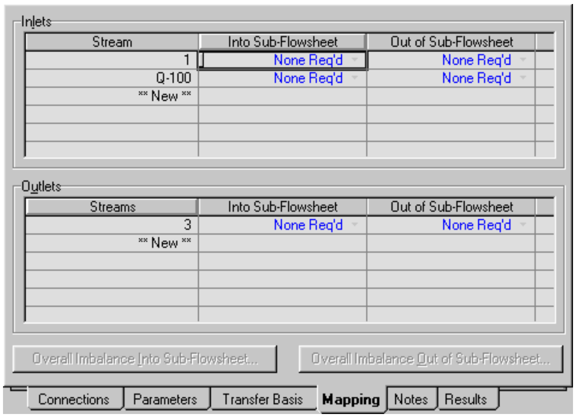
The transfer basis becomes significant only when the subflowsheet and Parent flowsheet fluid packages consist of different property methods.

13.2.5 Mapping Tab

Refer to **Chapter 6 - Component Maps** in the **HYSYS Simulation Basis** guide for more information.

The Mapping tab allows you to map fluid component composition across fluid package boundaries.

Figure 13.8



To attach a component map to inlet and outlet streams, specify the name of the inlet component map in the **In to Sub-Flowsheet** field and the name of the outlet component map in the **Out of Sub-Flowsheet** field of the desired stream.

Component Maps can be created and edited in the Basis environment.

Click the **Overall Imbalance Into Sub-Flowsheet** or **Overall Imbalance Out of Sub-Flowsheet** button to open the Untransferred Component Info property view. The Untransferred Component Info property view allows you to confirm that all of the components have been transferred into or out of the subflowsheet.

13.2.6 Notes Tab

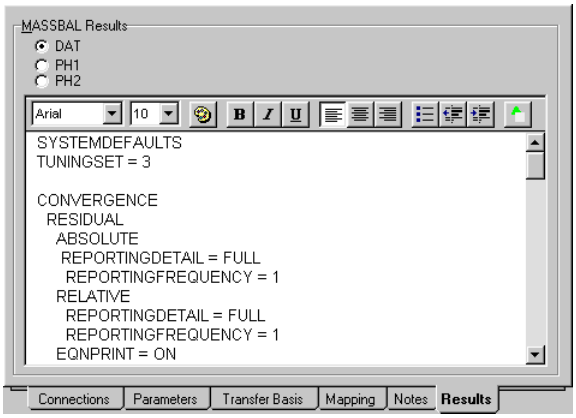
For more information, refer to **Section 1.3.5 - Notes Page/Tab**.

The Notes tab provides a text editor where you can record any comments or information regarding the material stream or your simulation case in general.

13.2.7 Results Tab

The calculated result from MASSBAL appears on the Results tab.

Figure 13.9

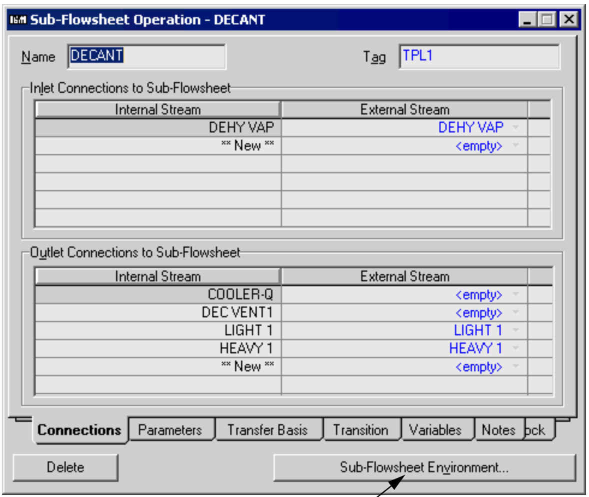


13.3 Subflowsheet Property View

The Subflowsheet property view consists of the following tabs:

- Connections
- Parameters
- Transfer Basis
- Mapping
- Variables
- Notes
- Lock

Figure 13.10



Click this button to enter the subflowsheet environment.

13.3.1 Adding a Subflowsheet

There are two ways you can add a subflowsheet to your simulation.

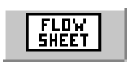
1. Select **Flowsheet | Add Operation** command from the menu bar. The UnitOps property view appears.

   You can also access the UnitOps property view by pressing **F12**.
2. Click the **Sub-Flowsheets** radio button.
3. From the list of available unit operations, select **Standard Sub-Flowsheet**.
4. Click the **Add** button.

OR

1. Select **Flowsheet | Palette** command from the menu bar. The Object Palette appears.

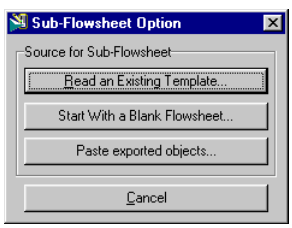
   You can also open the Object Palette by pressing **F4**.
2. Double-click on the **Sub-Flowsheet** icon on the Object Palette.



Sub-Flowsheet icon

The Sub-Flowsheet Option property view appears.

Figure 13.11



The Sub-Flowsheet Option property view contains the following options:

- Read an Existing Template
- Start with a Blank Flowsheet
- Paste exported objects
- Cancel

Read an Existing Template

For more information, refer to **Section 3.5.2 - Creating a Template Style Flowsheet** in the **HYSYS User Guide**.

If you want to use a previously constructed Template that has been saved on disk, click the Read an Existing Template button on the Sub-Flowsheet Option property view.

Start with a Blank Flowsheet

If you want to start with a blank subflowsheet, click the **Start with a Blank Flowsheet** button on the Sub-Flowsheet Option property view, HYSYS will install a subflowsheet operation containing no unit operations or streams.

On the **Connections** tab of the property view of the blank subflowsheet, there will be no feed or product connections (boundary streams) to the subflowsheet. You can connect feed streams in the External Stream column by either typing in the name of the stream to create a new stream or selecting a pre-defined stream from a drop-down list. When an external feed connection is made by selecting a pre-defined stream from the drop-down list, a stream similar to the pre-defined stream is created inside the subflowsheet environment.

In order to fully define the flowsheet, you have to enter the subflowsheet environment. Click the **Sub-Flowsheet Environment** button on the property view to transition to the subflowsheet environment and its dedicated Desktop. The subflowsheet is constructed using the same methods as the main flowsheet. When you return to the Parent environment, you can connect the subflowsheet boundary streams to streams in the Parent flowsheet.

Paste Exported Objects

If you want to import previously exported objects into a new subflowsheet, click the **Paste Exported Objects** button on the Sub-Flowsheet Option property view.

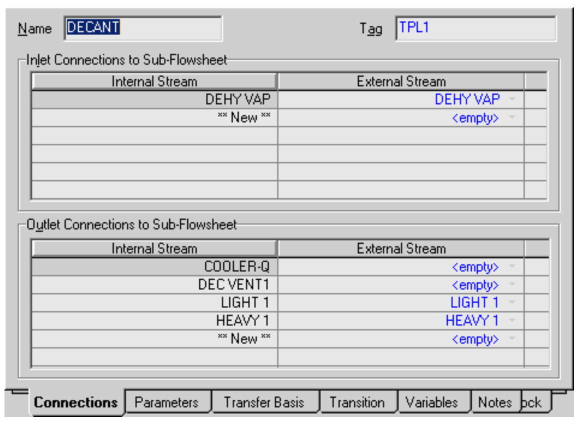
The objects that are selected and exported via the PFD can be imported back into a flowsheet without creating a new subflowsheet first.

You copy and paste selected objects inside the same subflowsheet or another subflowsheet. You can also copy and paste subflowsheets and column subflowsheets. Objects can also be moved into or out of a subflowsheet.

13.3.2 Connections Tab

You can enter the name of the subflowsheet, as well as its Tag name, on the Connections tab. All feed and product connections appear on the Connections tab.

Figure 13.12



Flowsheet Tags

These short names are used by HYSYS to identify the flowsheet associated with a stream or operation when that flowsheet object is being viewed outside of its native flowsheet scope. The default Tag name for a subflowsheet operation is TPL1 (for Template).

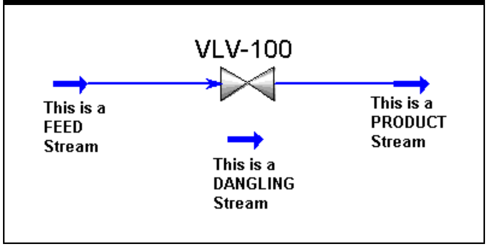
When more than one subflowsheet operation is installed, HYSYS ensures unique tag names by incrementing the numerical suffix; the subflowsheets are numbered sequentially in the order they were installed. For example, if the first subflowsheet added to a simulation contained a stream called Comp Duty, it would appear as Comp Duty@TPL1 when viewed from the Main flowsheet of the simulation.

Feed and Product Connections

Internal streams are the boundary streams within the subflowsheet that can be connected to external streams in the Parent flowsheet. Internal streams cannot be specified on this tab, they are automatically determined by HYSYS. Basically, any streams in the subflowsheet that are not completely connected (in other words,"open ended") can serve as a feed or product.

Subflowsheet streams that are not connected to any unit operations in the subflowsheet appear in the property view as well (and are termed “dangling streams”).

Figure 13.13



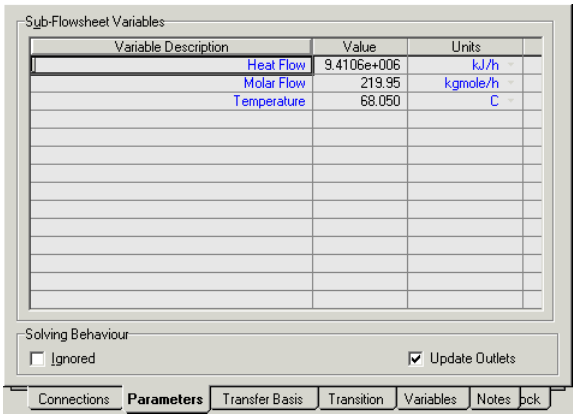
To connect the subflowsheet, specify the appropriate name of the external streams, which are in the Parent flowsheet, in the matrix opposite the corresponding internal streams, which are in the subflowsheet. The stream conditions are passed across the flowsheet boundary via these connections.

It is not necessary to specify an external stream for each internal stream.

13.3.3 Parameters Tab

You can view the exported subflowsheet variables on the Parameters tab, which allows you to keep track of several key variables without entering the subflowsheet environment or adding the variables to the global DataBook. It is also useful when dealing with a subflowsheet as a **black box**. The user who created the subflowsheet can set up an appropriate Parameters tab, and another user of the subflowsheet can be unaware of the complexities within the subflowsheet.

Figure 13.14



These variables display values, which have been calculated or specified by the user. If changes to the specified values are made here, the subflowsheet is updated accordingly. For each variable, the description, value, and units are shown.

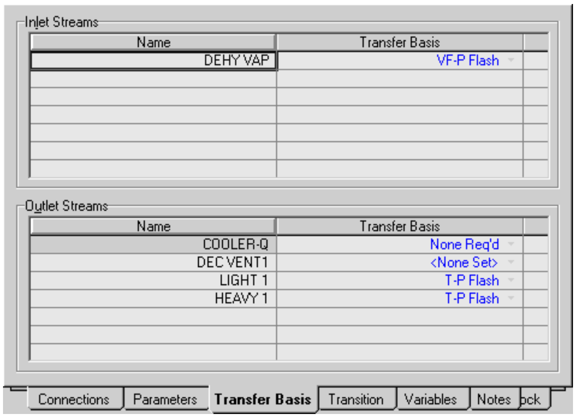
These variables are actually added on the Variables Tab of the property view, but are viewed in full detail on the Parameters tab.

- The Ignore checkbox is used to bypass the subflowsheet during calculations, just as with all HYSYS unit operations.
- The Update Outlets checkbox enables you to toggle between transferring or not transferring updated values from the subflowsheet to the external streams.

13.3.4 Transfer Basis Tab

The transfer basis for each Feed and Product Stream is listed on the Transfer Basis tab.

Figure 13.15



The Transfer Basis is also useful in controlling VF, T, or P calculations in column subflowsheet boundary streams with close boiling or nearly pure compositions.

The transfer basis only becomes significant when the subflowsheet and Parent flowsheet's fluid packages consist of different property methods. The transfer basis is used to provide a consistent means of switching between the different basis of the various property methods. The table below list all the possible transfer basis provided by HYSYS:

Transfer Basis	Description
T-P Flash	The Pressure and Temperature of the Material stream are passed between flowsheets. A new Vapour Fraction is calculated.
VF-T Flash	The Vapour Fraction and Temperature of the Material stream are passed between flowsheets. A new Pressure is calculated.
VF-P Flash	The Vapour Fraction and Pressure of the Material stream are passed between flowsheets. A new Temperature is calculated.

Transfer Basis	Description
P-H Flash	The Pressure and Enthalpy of the Material stream are passed between flowsheets.
User Specs	You define the properties passed between flowsheets for a Material stream.
None Required	No calculation is required for an Energy stream. The heat flow is simply passed between flowsheets.
<None Set>	No transfer basis has been selected.

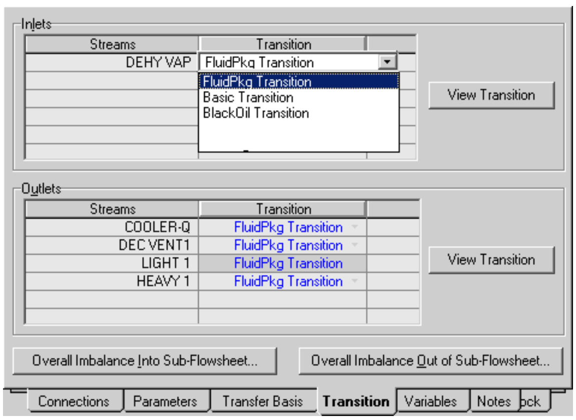
13.3.5 Transition Tab

The Transition tab allows you to select and modify the stream transfer and map methods for the fluid component composition across fluid package boundaries.

You may choose between three transition types: FluidPkg Transition, Basis Transition, and Black Oil Transition.

Fluid Package Transition

Figure 13.16

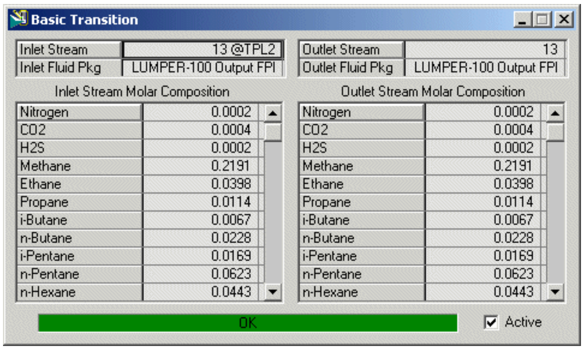


Composition values for individual components from one fluid package can be mapped to a different component in an alternate fluid package. Mapping is especially useful when dealing with hypothetical oil components where like components from one fluid package can be mapped across the subflowsheet boundary to another fluid package.

Component Maps can also be created and edited in the Basis environment.

Basic Transition

Figure 13.17

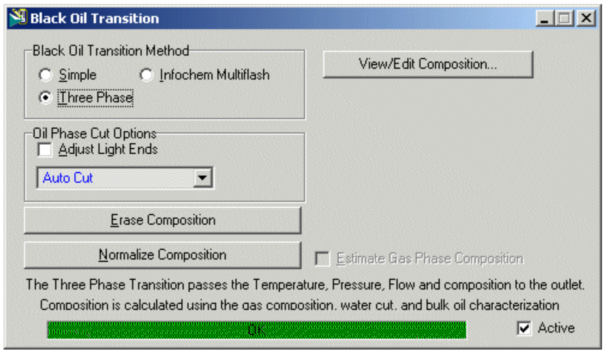


The Basic Transition view outlines the value of each component within the Inlet Stream Molar Composition and the Outlet Stream Molar Composition.

Refer to **Section 6.2 - Component Maps Tab** in the **HYSYS Simulation Basis** guide for more information.

Black Oil Transition

Figure 13.18



The Black Oil Transition view allows you to:

- Choose between three sperarate Black Oil Transition Methods:
  - Simple
  - Three Phase
  - Infochem Mulitflash
- Select the Oil Phase Cut Options from the drop-down list.
- View/Edit the Composition of the stream.
- Erase the Compostion of the stream.
- Normalize the Composition of the stream.

For every pairing of different fluid packages, a collection of maps exists. Component maps can be added to each collection on the Component Maps tab in the Simulation Basis Manager property view.

To select a transfer and map method for the inlet and outlet streams:

1. In the approriate cell under the **Transition** column, click the

down arrow icon to open the drop-down list.

2. In the drop-down list, select the transition type you want to apply to the stream.

> **Some of the transition method require the RefSYS or Upstream license to run.**

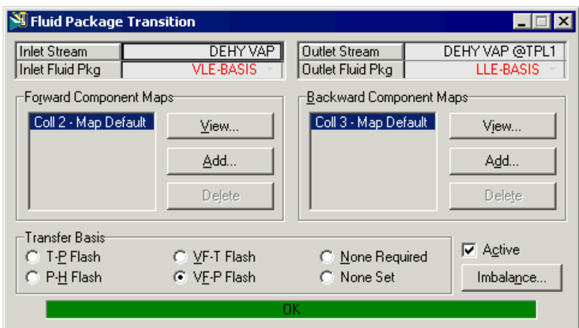
3. Repeat the above steps for all the streams that require a transition method.

To modify the type of transfer and map method for inlet and outlet streams:

1. Under the **Stream** column, click on the cell containing the stream you want to modify.
2. Click the appropriate **View Transition** button in the group.

   The Transition property view of the selected stream appears.

Figure 13.19



3. In the Transition property view, you can make the following changes:
   - modify the fluid package of the streams
   - edit, add, or delete a component map method
   - modify the transfer basis
4. Click the **Imbalance** button to view the component imbalance in the selected stream.

To view overall component imbalance for streams flowing through the subflowsheet:

1. Click the **Overall Imbalance Into Sub-Flowsheet** button

or **Overall Imbalance Out of Sub-Flowsheet** button to open the Untransferred Component Info property view.

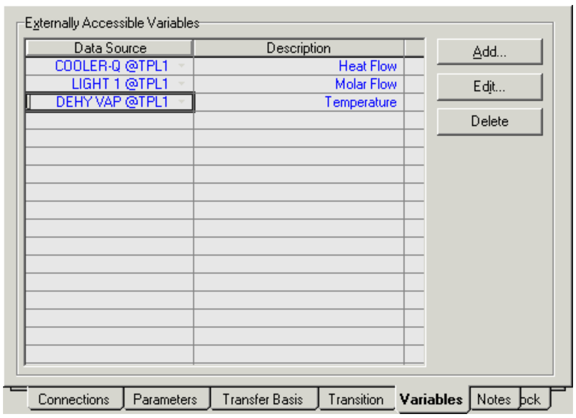
The Untransferred Component Info property view allows you to confirm that all of the components have been transferred into or out of the subflowsheet.

2. Click the appropriate radio button to view the imbalance in molar, mass, or liquid volume basis.

13.3.6 Variables Tab

The Variables tab of the Main flowsheet property view allows you to create and maintain the list of externally accessible variables.

Figure 13.20



Although you can access any information inside the subflowsheet using the Variable Navigator, the features on the Variables tab allow you to target key process variables inside the subflowsheet and display their values on the property view. Then, you can conveniently view this whole group of information directly on the subflowsheet property view in the Parent flowsheet.

Refer to **Section 1.3.9 - Variable Navigator Property View** for information on the Variable Navigator.

To add variables:

1. Click the **Add** button. The Variable Navigator property view

appears.

2. On the Variable Navigator property view, select the flowsheet object and variable you want.

   You can also over-ride the default variable description displayed in the Variable Description field of the Variable Navigator property view.

> **Any subflowsheet variables added in the Variables tab will appear on the Parameters tab.**

13.3.7 Notes Tab

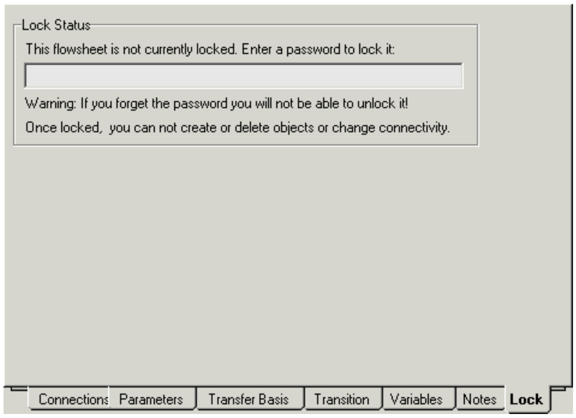
For more information, refer to **Section 1.3.5 - Notes Page/Tab**.

The Notes tab provides a text editor where you can record any comments or information regarding the material stream or to your simulation case in general.

13.3.8 Lock Tab

The Lock tab enables you to lock or unlock the subflowsheet and displays the lock status of the subflowsheet.

Figure 13.21



When the flowsheet is locked, you cannot create or delete objects, or change the topology. You can add Set, Adjust, and Spreadsheet operations; manipulate variable values; or copy the contents of the flowsheet and create your own modifiable version.

Subflowsheets inside a locked subflowsheet have to be specifically locked.

- To lock a subflowsheet, enter a password in the **Lock Status** field and press **ENTER**.
- To unlock a subflowsheet, enter the correct password in the **Lock Status** field and press **ENTER**.

14 Utilities

14.1 Introduction

For information on adding the utilities using the Available Utilities property view, refer to the section on **Adding a Utility**.

The utility commands are a set of tools, which interact with a process by providing additional information or analysis of streams or operations. In HYSYS, utilities become a permanent part of the Flowsheet and are calculated automatically when appropriate. They can also be used as target objects for Adjust operations.

Most utilities can also be added through the Utilities page on the Attachments tab of a stream's property view. A utility added through either route is automatically updated in the other location. For example, if you attach an Envelope utility to a stream using the Available Utilities property view, the Envelope utility automatically appears on the Utilities page of the Attachments tab in the property view of the stream to which it was attached.

You can select any of the following utilities from the Available Utilities property view:

For further details, refer to **Chapter 1** in the **Aspen RTO Reference Guide**.

For further details, refer to **Chapter 1** in the **Aspen RTO Reference Guide**.

Utilities	Description
Boiling Point Curves	Obtains laboratory-style distillation results for streams.
CO2 Freeze Out	Determines stream CO2 freezing conditions.
Cold Properties	Calculates several stream Cold Properties, for example True and Reid Vapour Pressures, Flash Point, Pour Point, Refractive Index, and so forth.
Composite Curves	Optimizes the use of process heat exchange and utilities for heat exchangers, LNG's, coolers, and heaters.
Critical Properties	Calculates true and pseudo critical properties for streams.
Data Recon	Used by HYSYS.RTO optimization objects as a data holder that allows for multiple sets of stream data, each corresponding to a different set.
Depressuring-Dynamics	Models the pressure letdown of a single vessel or network of vessels under plant emergency conditions.
Derivative	Used by HYSYS.RTO to hold all the data used for defining the RTO optimizer constraints and variables.
Envelope	Shows critical values and phase diagrams for a stream.

Utilities	Description
Hydrate Formation	Determines stream hydrate formation conditions.
Parametric	The Parametric Utility integrates Neural Network (NN) technology into its framework. The major function of the utility is to approximate an existing HYSYS model with a parametric model.
Pipe Sizing	Performs design calculations on any of the case streams.
Property Balance	Performs balance calculations across any utilities. You can select individual utilities or the entire flowsheet.
Property Table	Examines stream property trends over a range of conditions.
Tray Sizing	Size or rate existing sections or full towers.
User Property	Defines new stream properties based on composition.
Vessel Sizing	Size and cost installed Separator Unit Operations.

Adding a Utility

1. Select **Tools | Utilities** command from the menu bar. The Available Utilities property view appears.

   You can also access the Available Utilities property view by pressing **CTRL U**.
2. From the list of available utilities, in the right pane, select the utility you want to add.
3. Click the **Add Utility** button. The selected utility's property view appears.

Editing a Utility

1. Select **Tools | Utilities** command from the menu bar. The Available Utilities property view appears.
2. From the list of installed utilities, in the left pane, select the utility you want to view.
3. Click the **View Utility** button. The selected utility's property view appears. From here, you can modify any of the utility's properties.

Deleting a Utility

1. Select **Tools | Utilities** command from the menu bar. The Available Utilities property view appears.
2. From the list of installed utilities, in the left pane, select the utility you want to delete.
3. Click the **Delete Utility** button. HYSYS will ask you to confirm the deletion.

You can also delete a utility by clicking the Delete button on the utility's property view.

Ignoring a Utility

To ignore a utility during simulation calculations:

1. Select **Tools | Utilities** command from the menu bar. The Available Utilities property view appears.
2. From the list of installed utilities, in the left pane, select the utility you want to view.
3. Click the **View Utility** button. The selected utility's property view appears.
4. Select the **Ignored** checkbox, which is usually located on right bottom corner of the utility's property view.

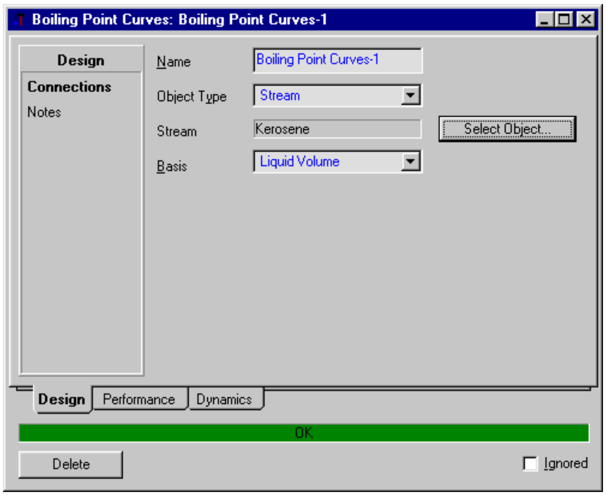
   HYSYS disregards the utility entirely until you restore the utility to an active state by clearing the **Ignored** checkbox.

14.2 Boiling Point Curves

Refer to **Chapter 4 - HYSYS Oil Manager** in the **HYSYS Simulation Basis** guide for details on the distillation data types.

The Boiling Point Curves utility, which generally is used in conjunction with characterized oils from the Oil Manager, allows you to obtain the results of a laboratory style analysis for your simulation streams. Simulated distillation data including TBP, ASTM D86, D86 (Corr.), D1160(Vac), D1160(Atm), and D2887 as well as critical property data for each cut point and cold property data are calculated. The data can be viewed in a tabular format or graphically.

Figure 14.1



To add the Boiling Point Curves utility, refer to the section on **Adding a Utility**.

The object for the analysis can be a stream, a phase on any stage of a tray section, or one of the phases in a separator, in a condenser or in a reboiler. You select the basis for the calculations, and you can specify the boiling ranges for the simulated distillation data.

14.2.1 Design Tab

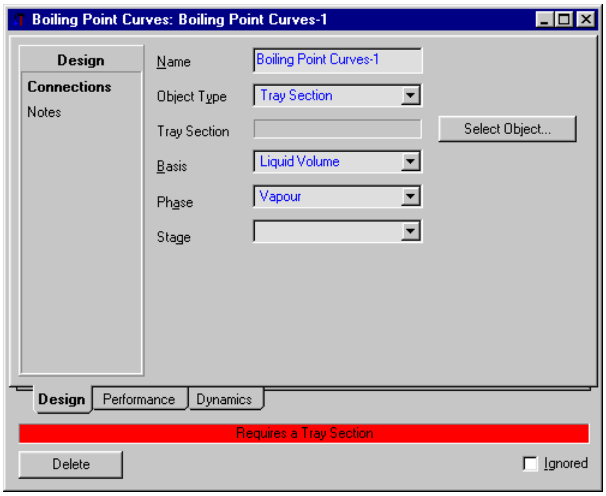
The Design tab contains the following pages:

- Connections
- Notes

Connections Page

On the Connections page, you can select the parameters for the Boiling Point Curves utility.

Figure 14.2



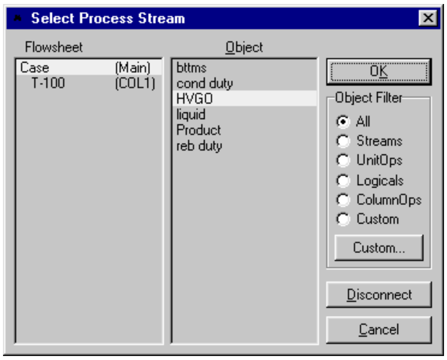
Setting the Utility Parameters

1. On the **Connections** page of the **Design** tab, change the Name of the utility, if desired.
2. From the **Object Type** drop-down list, select the object type you want. The options are Stream, Tray Section, Separator, Condenser, or Reboiler.

   For a **tray section**, the boiling point curves and critical property data can be accessed on the Profiles tab of the Column Runner.

3. Click the **Select Object** button, the Select (object type) view appears.

Figure 14.3



The title of the Select (object type) view depends on the object type you selected. For example, if you select the condenser, the Select Condenser property view appears.

4. Choose the appropriate object from the Object list, and click the **OK** button to add the selected object to the utility.

   The Object list can be filtered by selecting one of the radio buttons in the Object Filter group.

5. From the Basis drop-down list, select the basis for the calculation of the distillation data. The options are Mole Frac, Mass Frac, Liquid Volume.
6. For all object types except the Stream selection, from the Phase drop-down list you can select the phase for the analysis as either Vapour or Liquid.
7. If the Object Type which you have selected is a Tray Section, from the Stage drop-down list select a stage.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor, where you can record any comments or information regarding the utility, or to your simulation case in general.

14.2.2 Performance Tab

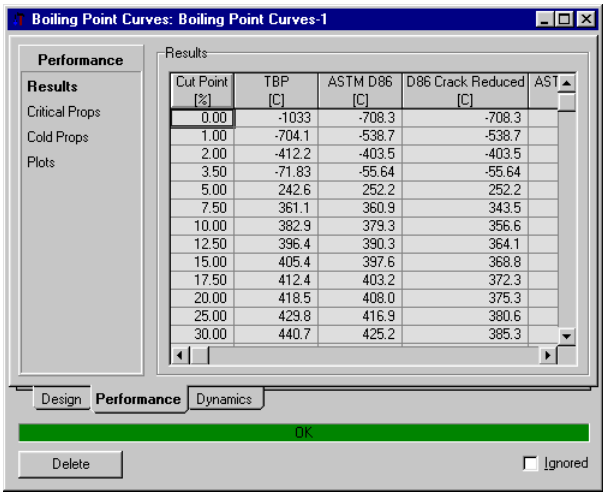
The Performance tab contains the following pages:

- Results
- Critical Props
- Cold Props
- Plots

Results Page

You can view the results of the boiling point curve calculations in tabular format on the Results page.

Figure 14.4



Simulated distillation profiles are provided for the following assay types:

- TBP
- ASTM D86
- D86 Corr.
- ASTM D1160 (Vac.)
- ASTM D1160 (Atm.)
- ASTM D2887

The ASTM D86 boiling point curve corresponds to the true boiling points of the oil, which assumes no cracking has occurred.

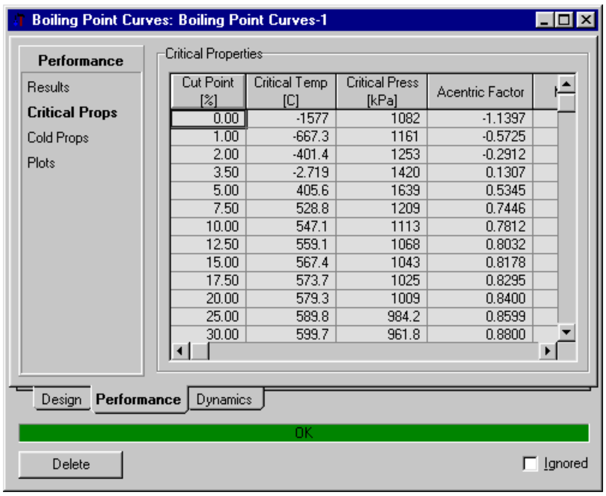
When the oil is characterized by a ASTM D86 distillation assay with no cracking option, the D86 Corr boiling point curve corresponds to the assay input data. The ASTM D86 boiling point curve then corresponds to raw lab data, with no cracking correction applied.

When the oil is characterized by a ASTM D86 distillation assay with cracking option, the ASTM D86 boiling point curve corresponds to the assay input data. The cracking correction factor is then applied to the D86 Corr boiling point curve.

Critical Props Page

The Critical Props page contains, for each cut point, the critical temperature, critical pressure, acentric factor, molecular weight, and liquid density.

Figure 14.5

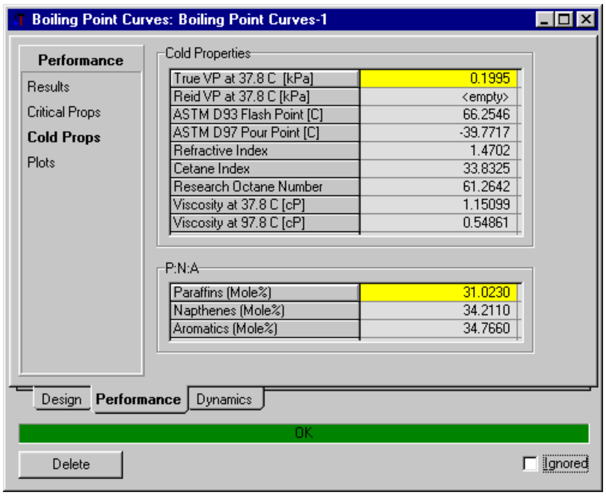


Cold Props Page

Details of the methods used to determine the Cold Properties can be found in **Section 14.4 - Cold Properties**.

You can view the bulk cold properties of the stream on the Cold Props page. Also listed is the ratio of paraffins to naphthas to aromatics.

Figure 14.6



Plots Page

The Plots page shows the Boiling Point Curves results and the Critical Properties results in graphical form. Examine the plot of your choice by making a selection from the Dependent Variable drop-down list:

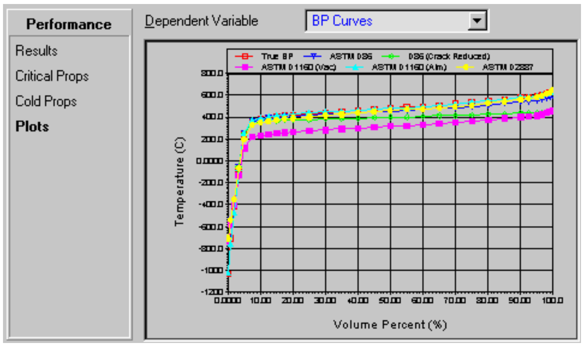
- Boiling Point Curves
- Critical Temperature
- Critical Pressure
- Acentric Factor
- Molecular Weight
- Liquid Density

Refer to **Section 1.3.1 - Graph Control Property View** for more information.

You can customize a plot by right-clicking in the plot area, and selecting **Graph Control** command from the object inspect menu.

The figure below shows an example of the Plots page.

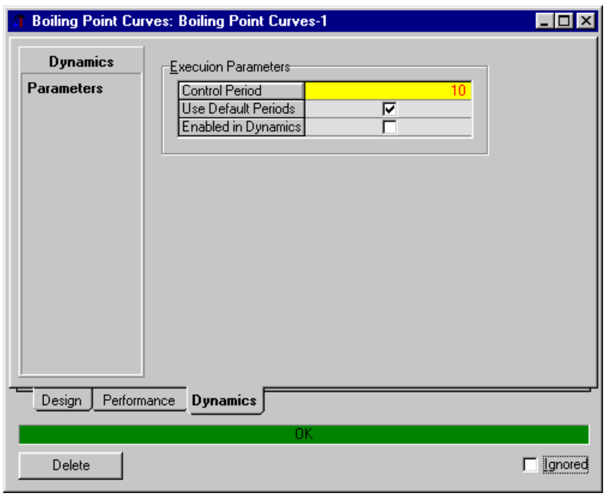
Figure 14.7



14.2.3 Dynamics Tab

The Dynamics tab allows you to control how often the utility gets calculated when running in Dynamic mode.

Figure 14.8



The Control Period field is used to specify the frequency that the utility is calculated. A value of 10 indicates that the utility be recalculated every 10th pressure flow step. This can help speed up your dynamic simulation since utilities can require some time to calculate.

The **Use Default Periods** checkbox allows you to set the control period of one utility to equal the control period of any other utilities that you have in the simulation. For example, if you have five utilities, and require them all to have a control period of 5 and currently the value is 8, with this checkbox selected if you change the value in one utility all the other utilities change. Alternatively, if you want all the utilities to have different values you would clear this checkbox.

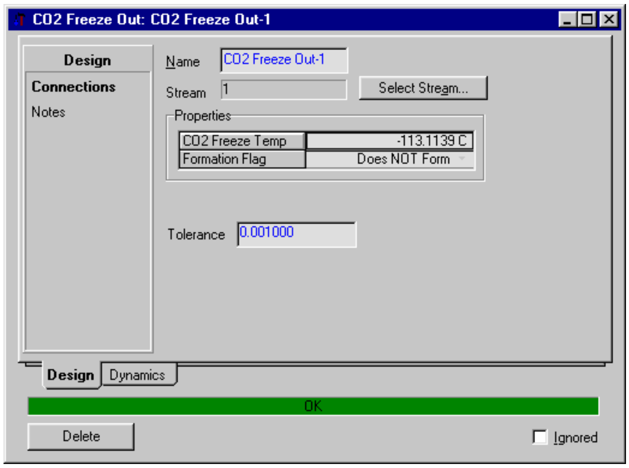
The **Enable in Dynamics** checkbox is used to activate this feature for use in Dynamic mode.

14.3 CO2 Solids

To add the CO2 Freeze Out utility, refer to the section on **Adding a Utility**.

An equation-of-state based approach is used to calculate the incipient solid formation point for mixtures containing Carbon Dioxide (CO2). The model can be used for predicting the initial solid formation point in equilibrium with either vapours or liquids. The fugacity of the resultant solid is obtained from the known vapour pressure of solid CO2. The fugacity of the corresponding phase (in equilibrium with the solid) is calculated from the equation of state.

Figure 14.9



CO2 Solids prediction is restricted to the Peng Robinson (PR) and Soave Redlich Kwong (SRK) equations of state.

14.3.1 Design Tab

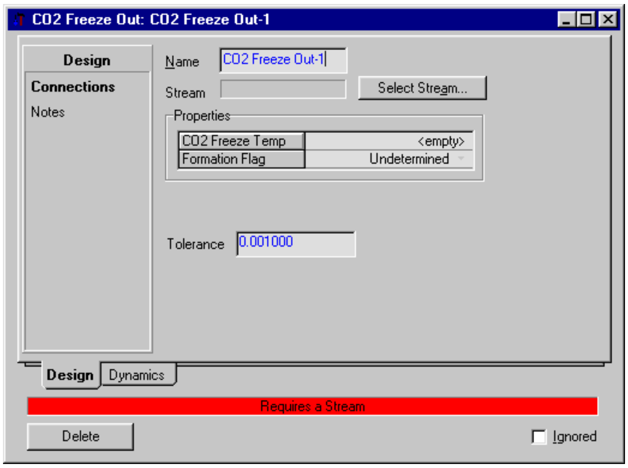
The Design tab contains the following pages:

- Connections
- Notes

Connections Page

You are required to specify the stream for which the calculations are made on the Connections page.

Figure 14.10



You can select the stream from the Select Process Stream property view, which is accessed by clicking the Select Stream button.

HYSYS determines the CO_2 Freeze Temperature, and displays the formation status in the Formation Flag field:

Formation Flag	Flag Significance
Undetermined	No Stream has been chosen.
NO CO2 in Stream	There is no CO_2 present in the Stream.

Formation Flag	Flag Significance
Does NOT Form	Solid CO_2 not form at the present conditions of the stream. The CO_2 Freeze Temperature is shown in the corresponding field.
Solid CO2 Present	Solid CO_2 is present at the current stream conditions. The CO_2 Freeze Temperature is shown in the corresponding field.

In the Tolerance field, you can specify the tolerance used to calculate the incipient solid formation point.

Notes Page

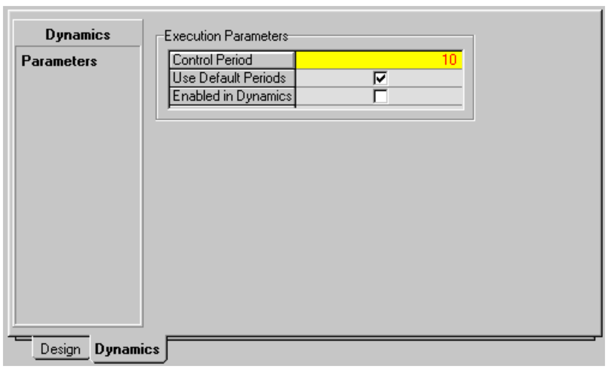
For more information, refer to **Section 1.3.5 - Notes Page/Tab**.

The Notes page provides a text editor, where you can record any comments or information regarding the utility, or to your simulation case in general.

14.3.2 Dynamics Tab

The Dynamics tab allows you to control how often the utility gets calculated when running in Dynamic mode.

Figure 14.11



The Control Period field is used to specify the frequency that the utility is calculated. A value of 10 indicates that the utility be recalculated every 10th pressure flow step. This can help speed up your dynamic simulation since utilities can require some time to calculate.

The **Use Default Periods** checkbox allows you to set the control period of one utility to equal the control period of any other utilities that you have in the simulation. For example, if you have five utilities, and require them all to have a control period of 5 and currently the value is 8, with this checkbox selected if you change the value in one utility all the other utilities change. Alternatively, if you want all the utilities to have different values you would clear this checkbox.

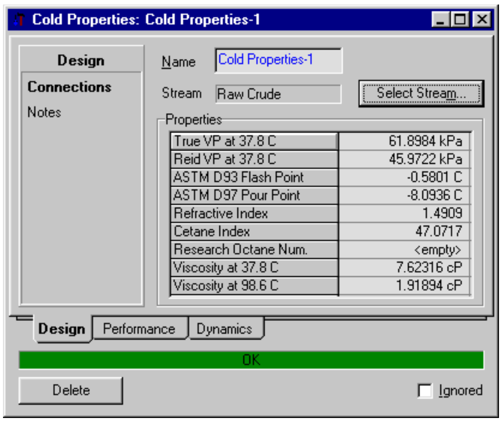
The Enable in Dynamics checkbox enables the Use Default Periods feature for use in Dynamic mode.

14.4 Cold Properties

To add the Cold Properties utility, refer to the section on **Adding a Utility**.

The Cold Properties utility enables you to view the cold properties of a stream.

Figure 14.12



The following list summarizes the cold properties which are available through the Cold Properties utility:

Cold Property	Calculations	Range of Validity
True Vapour Pressure @ 100°F (37.8°C)	Vapour Pressure method of selected property package	P>1.5 kPa
Reid Vapour Pressure @ 100°F (37.8°C)	Vapour pressure of system when vapour:liquid ratio by volume is 4:1	P>1.5 kPa

Cold Property	Calculations	Range of Validity
Flash Point	As per API 2B7.1	150°F<ASTM D86 10% (or NBP)<1150°F, -15°F<Flash Point<325°F
Pour Point	As per API 2B8.1	140<MW<800,1<API gravity<50, -110°F<Flash Point<140°F
Refractive Index	As per API 2B5.1-1	70<MW<600, 97°F<NBP<1000°F, 0.63<sg<1.1, 1.35<Refractive Index at 20°C<1.65
Cetane Index (Diesel Index)	Proprietary method	300°F<D86 10%<700°F
Research Octane Number (R.O.N.)	Proprietary method	D86 50% ~420°F
Viscosity at 100°F (37.8°C)	Refer to the **Viscosity** section in **Appendix A - Property Methods & Calculations** in the **HYSYS Simulation Basis** guide.	
Viscosity at 210°F (98.6°C)	Refer to the **Viscosity** section in **Appendix A - Property Methods & Calculations** in the **HYSYS Simulation Basis** guide.	
ASTM D86 Distillation Curve	API Figure 3A1.1 (1963)	51°F<TBP 10%<561°F
P/N/A (mol%)	As per API 2B4.1	MW>70

14.4.1 Design Tab

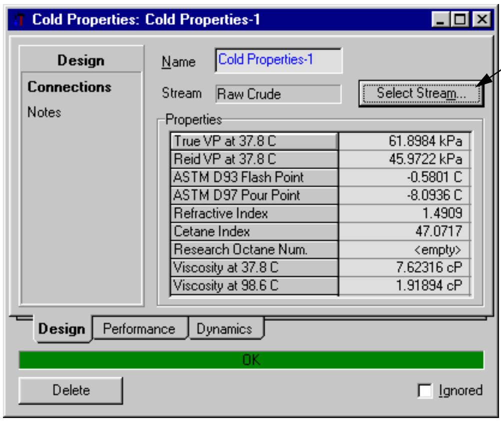
The Design tab contains the following pages:

- Connections
- Notes

Connections Page

You can attach a stream to the utility, and view the streams properties on the Connections page.

Figure 14.13



Click the **Select Stream** button to open the Select Stream Process property view.

In the Select Stream Process property view, you can select the stream you want.

The Properties group displays the following properties:

- True Vapour Pressure
- Reid Vapour Pressure
- Flash Point
- Pour Point
- Refractive Index
- Cetane Index
- Research Octane Number
- Viscosity at 100°F (37.8°C) and 210°F (98.6°C)

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor, where you can record any comments or information regarding the utility, or to your simulation case in general.

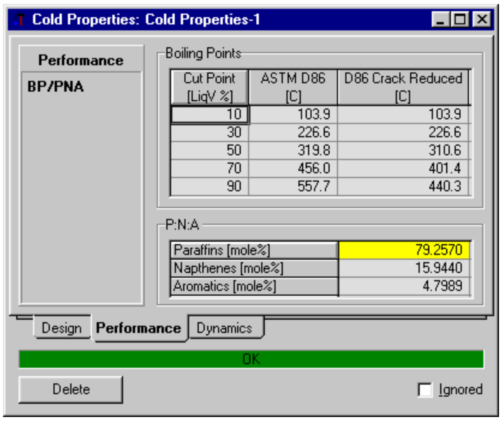
14.4.2 Performance Tab

The Performance tab contains only the BP/PNA page.

BP/PNA Page

The BP/PNA page displays the ASTM Distillation Curve (ASTM D86 10%, 30%, 50%, 70%, 90% Points), and the P/N/A mole percents.

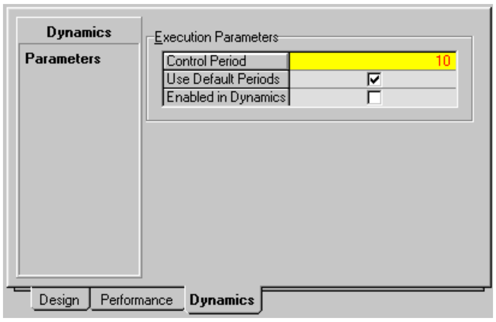
Figure 14.14



14.4.3 Dynamics Tab

The Dynamics tab allows you to control how often the utility gets calculated when running in Dynamic mode.

Figure 14.15



The Control Period field is used to specify the frequency that the utility is calculated. A value of 10 indicates that the utility be recalculated every 10th pressure flow step. This can help speed up your dynamic simulation since utilities can require some time to calculate.

The **Use Default Periods** checkbox allows you to set the control period of one utility to equal the control period of any other utilities that you have in the simulation. For example, if you have five utilities, and require them all to have a control period of 5 and currently the value is 8, with this checkbox selected if you change the value in one utility all the other utilities change. Alternatively, if you want all the utilities to have different values you would clear this checkbox.

The **Enable in Dynamics** checkbox is used to activate this feature for use in Dynamic mode.

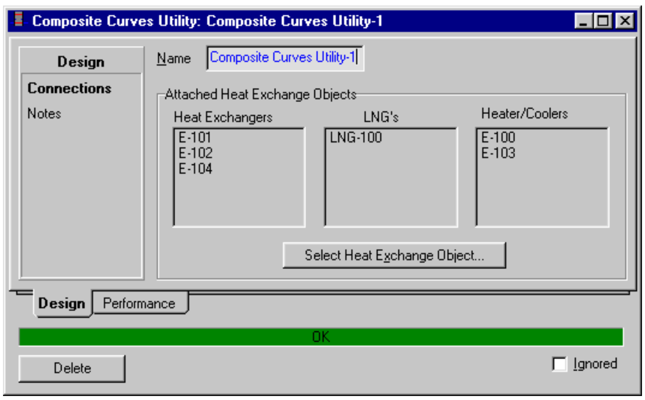
14.5 Composite Curves Utility

To add the Composite Curves utility, refer to the section on **Adding a Utility**.

Pinch technology is a methodology, which is used to optimize the use of process heat exchange and utilities in complicated processes. The HYSYS Composite Curves utility provides the necessary tools to apply the pinch principles in the design of efficient heat exchanger networks. For further pinch analysis information, refer to the text by Marsland[1].

You can attach any combination of heat exchangers, LNG operations, heaters or coolers to the Composite Curves utility. The only requirement being that each operation is solved so the Pinch calculations can be performed.

Figure 14.16



14.5.1 Design Tab

The Design tab contains the following pages:

- Connections
- Notes

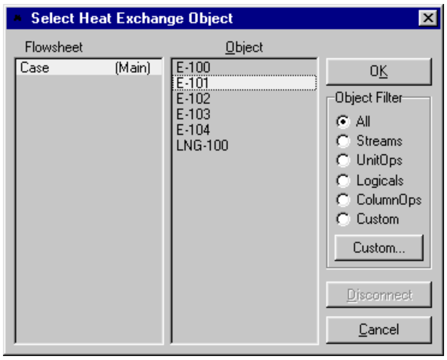
Connections Page

On the Connections page, you can attach any combination of heat exchangers, LNG operations, heaters, and coolers to the utility.

Adding a Heat Exchanger Object

1. On the **Connections** page of the **Design** tab, click the **Select Heat Exchanger Object** button.
2. The Select Heat Exchanger Object property view appears.

Figure 14.17



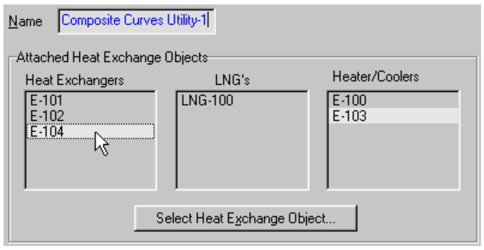
3. From the property view, select the heat exchanger, LNG, heater or cooler you want from the Object list.

   The Object list can be filtered by selecting one of the radio buttons in the Object Filter group.
4. Click the **OK** button to add the selected object to the utility.

Removing a Heat Exchanger Object

1. On the **Connections** page of the **Design** tab, select the heat exchanger object you want to remove from the list.

Figure 14.18



2. Press the **DELETE** button.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor, where you can record any comments or information regarding the utility, or to your simulation case in general.

14.5.2 Performance Tab

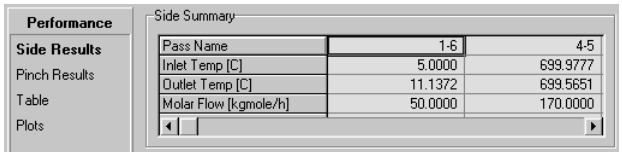
The Performance tab contains the following pages:

- Side Results
- Pinch Results
- Table
- Plots

Side Results Page

On the Side Results page, you can view the inlet temperature, outlet temperature, and molar flow of each pass attached to the Composite Curves utility.

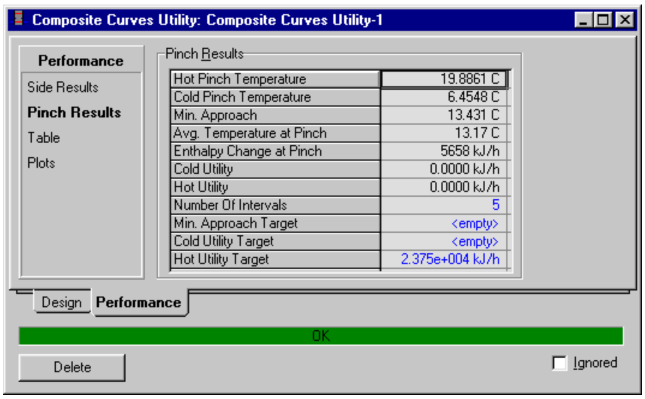
Figure 14.19



Pinch Results Page

The various results of the Composite Curves utility can be examined on the Pinch Results page.

Figure 14.20



The results which can be examined include the following:

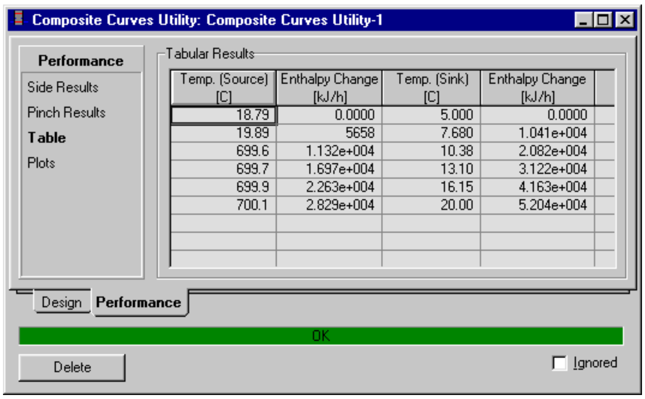
- Hot Pinch Temperature
- Cold Pinch Temperature
- **Minimum Approach**. Temperature difference between the Hot Pinch and Cold Pinch.
- Average Temperature at Pinch
- Enthalpy Change at Pinch

- Cold Utility
- Hot Utility
- **Number of Points**. The number of intervals used in the Composite Curves utility calculations.
- **Minimum Approach Target**. Specifiable minimum approach temperature.
- **Cold Utility Target**. Specifiable cold utility enthalpy value.
- **Hot Utility Target**. Specifiable hot utility enthalpy value.

Table Page

The Table page shows a tabular report of what is seen on the Plots page. You can view temperatures of the Sink and Source, the LMTD and enthalpy change for each interval.

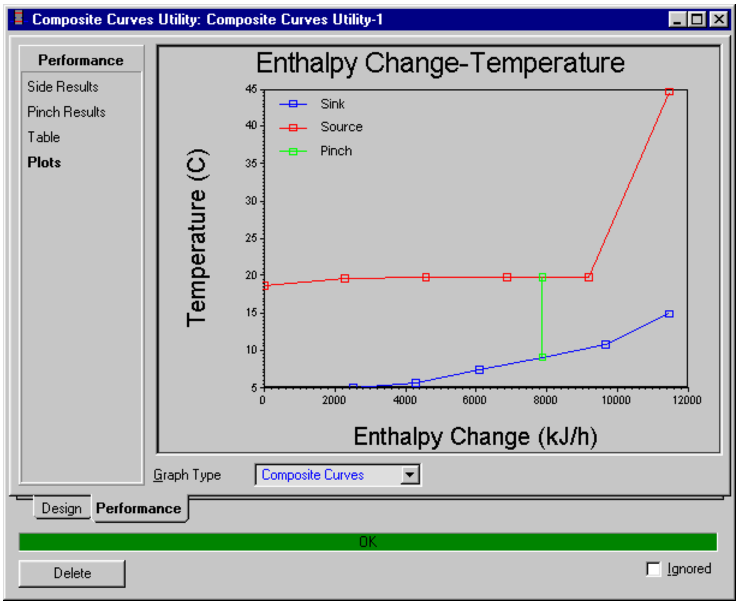
Figure 14.21



Plots Page

On the Plots page, you can view the Sink and Source Composite Curves or the Grand Composite Curve. Make your selection from the Graph Type drop-down list. The Composite Curves for the heat exchanger network is shown in the figure below. Notice the pinch also appears on this plot.

Figure 14.22



Refer to **Section 1.3.1 - Graph Control Property View** for more information about manipulating plots.

You can edit the plot by right-clicking anywhere in the plot area, and selecting the **Graph Control** command from the object inspect menu.

14.6 Critical Properties

To add the Critical Properties utility, refer to the section on **Adding a Utility**.

The Critical Properties utility calculates both the true and pseudo critical temperature, pressure, volume, and compressibility factor for a fully defined stream.

True & Pseudo Critical Properties

The Critical Properties utility displays two sets of critical properties, true and pseudo critical properties. **True Critical Properties** are those properties calculated using the mixing rules associated with the property package chosen. **Pseudo Critical Properties** use simple linear models to estimate the critical properties of a mixture. They are often very different from the true critical points and **have no real physical significance**, but sometimes are used in empirical correlations.

Mathematically, the pseudo critical temperature, pressure, and compressibility (T_{pc}, P_{pc}, *and* Z_{pc}) are defined as:

$$T_{pc} = \sum_{i=1}^{n} y_i T_{ci} \tag{14.1}$$

$$P_{pc} = \sum_{i=1}^{n} y_i P_{ci} \tag{14.2}$$

$$Z_{pc} = \sum_{i=1}^{n} y_i Z_{ci} \tag{14.3}$$

where:

y_i = *mole fraction of component i*

n = *total number of components in mixture*

T_{ci} = *critical temperature of component i*

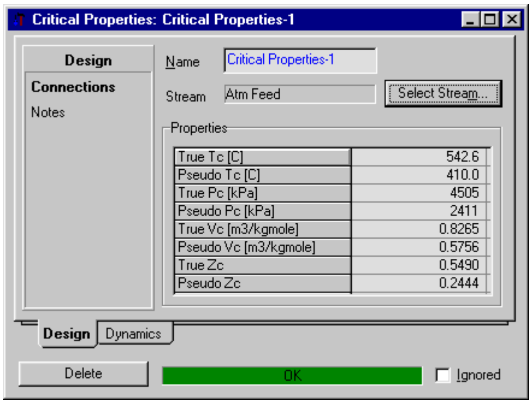
P_{ci} = *critical pressure of component i*

Z_{ci} = *critical compressibility of component i*

The remaining pseudo critical property, pseudo critical volume v_{pc}, is calculated using the following relationship:

$$v_{pc} = \frac{Z_{pc}T_{pc}R}{P_{pc}} \tag{14.4}$$

Figure 14.23



You must set up a fluid package using the Peng Robinson property method to use this utility.

14.6.1 Design Tab

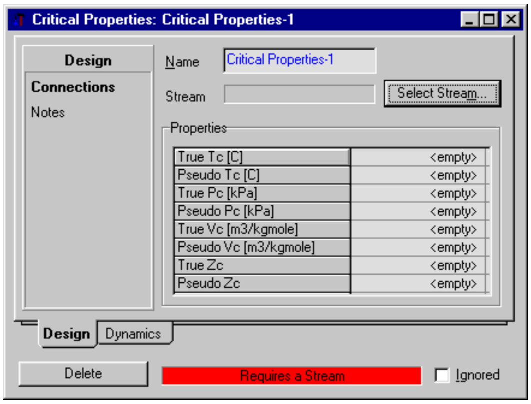
The Design tab contains the following pages:

- Connections
- Notes

Connections Page

You can connect the utility to a stream, and change the name of the utility on this page.

Figure 14.24



The following is the general procedure for connecting a stream to the Critical Properties utility:

1. On the Critical Properties property view, specify the name of the utility.
2. Click the **Select Stream** button. The Select Process Stream property view appears.
3. Select the stream you created from the Object list.

   The Object list can be filtered by selecting one of the radio buttons in the Object Filter group.
4. Click the **OK** button to add the selected stream to the utility.

Critical Property Analysis

You can examine the critical property values for the selected stream in the Properties group.

Notes Page

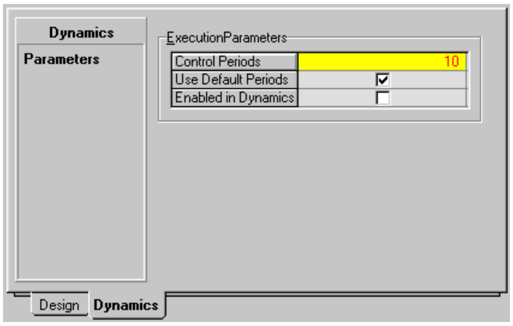
For more information, refer to **Section 1.3.5 - Notes Page/Tab**.

The Notes page provides a text editor where you can record any comments or information regarding the utility or your simulation case in general.

14.6.2 Dynamics Tab

The Dynamics tab allows you to control how often the utility gets calculated when running in Dynamic mode.

Figure 14.25



The Control Period field is used to specify the frequency that the utility is calculated. A value of 10 indicates that the utility be recalculated every 10th pressure flow step. This can help speed up your dynamic simulation since utilities can require some time to calculate.

The **Use Default Periods** checkbox allows you to set the control period of one utility to equal the control period of any other utilities that you have in the simulation. For example, if you have five utilities, and require them all to have a control period of 5 and currently the value is 8, with this checkbox selected if you change the value in one utility all the other utilities change. Alternatively, if you want all the utilities to have different values you would clear this checkbox.

The **Enable in Dynamics** checkbox is used to activate this feature for use in Dynamic mode.

14.7 Data Recon Utility

To add the Data Recon utility, refer to the section on **Adding a Utility**.

The Data Recon utility is a component of the HYSYS.RTO real-time optimization package available as a plug-in to the basic HYSYS software package. The Data Recon utility is one of two utilities used by HYSYS.RTO to provide the primary interface between the flowsheet model and the solver. Their primary purpose is to collect appropriate optimization objects which are then exposed to solvers to meet a defined solution criteria.

Refer to the **Chapter 6 - Data Reconciliation Utility** in the **Aspen RTO Reference Guide** for details concerning the use of this utility. This guide details all features and components related to the HYSYS real time optimization package.

If your current HYSYS version does not support RTO, contact your local AspenTech representative for more details.

14.8 Derivative Utility

To add the Derivative utility, refer to the section on **Adding a Utility**.

The Derivative utility is a component of the HYSYS.RTO real-time optimization package available as a plug-in to the basic HYSYS software package. The Derivative utility is one of two utilities used by HYSYS.RTO to provide the primary interface between the flowsheet model and the solver. Their primary purpose is to collect appropriate optimization objects, which are then exposed to solvers to meet a defined solution criteria.

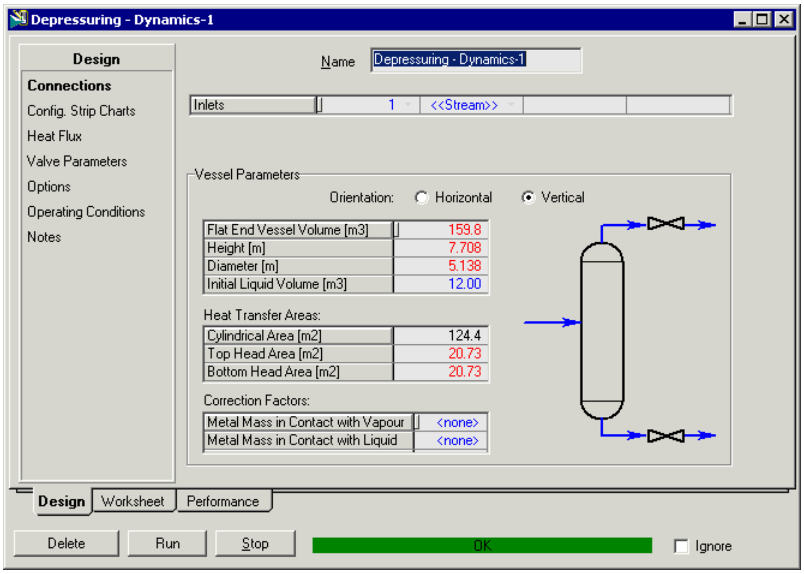
If your current HYSYS version does not support RTO, contact your local AspenTech representative for more details.

Refer to the **Aspen RTO Reference Guide** for details concerning the use of this utility. This guide details all features and components related to the HYSYS real time optimization package.

14.9 Dynamic Depressuring

The dynamic depressuring utility is mostly an interface where data is entered.

Figure 14.26

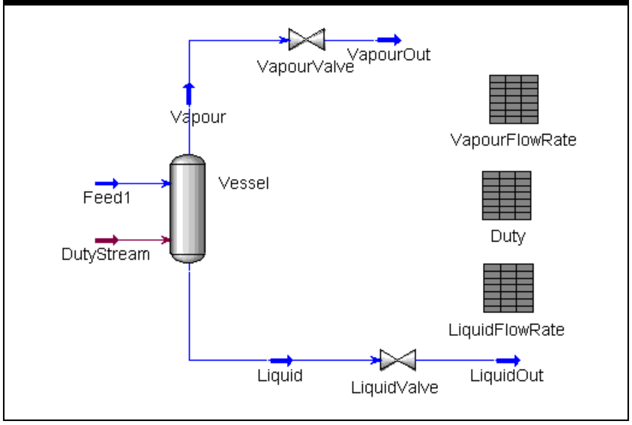


To start the Dynamic Depressuring calculations, specify enough information in the utility, and click the Run button on the Dynamic Depressuring property view. If you want to stop the utility while it is calculating, click the Stop button.

The dynamic depressuring utility does not require dynamics or other additional special licenses to run.

The entered data gets transferred to a subflowsheet of the depressuring system (one inlet stream and one vessel).

Figure 14.27



To add the Dynamic Depressuring utility, refer to the section on **Adding a Utility**.

It is this subflowsheet that is run in dynamics until the depressuring time is complete, and the system then returns to steady state. The results are retrieved from the strip charts, and displayed on the Performance tab.

The Dynamic Depressuring utility works with any fluid package, except for the electrolyte fluid package and where solids are present.

Dynamic Depressuring Subflowsheet

The dynamic depressuring subflowsheet is not meant to be altered in any way. In fact, before the utility runs HYSYS checks to see if the template has been changed (for example, if the number of streams and unit operations has change), and if it has been changed HYSYS deletes the altered subflowsheet and creates a new one.

There are three spreadsheets in the subflowsheet:

- vapour flow rate
- liquid flow rate
- duty

When the utility runs, the values of the variables for the selected equation are transferred to the spreadsheet.

The three spreadsheets are used for the flow rate and heat flux equations. The calculated flows are exported from the spreadsheets to either the vapour, liquid, or duty stream flow.

The spreadsheets are used to transfer data from the utility, which is manipulated and then sent to different unit operations. As a result the spreadsheet is being over written every time the utility runs.

If you modify the spreadsheet and run the utility, your added information is lost. The only time a spreadsheet is not overwritten is when the Use Spreadsheet option / mode is selected.

The Dynamic Depressuring utility provides an **Use Spreadsheet** option for both the liquid and vapour flow rate, and the heat flux equation. The Use Spreadsheet option gives unlimited possibilities of flow rate and heat flux equations.

You can select the Use Spreadsheet option from the Operating Mode drop-down list on the **Heat Flux** page of the **Design** tab in the Dynamic Depressuring utility property view. When you select the Use Spreadsheet option, a **View Spreadsheet** button appears. By clicking the **View Spreadsheet** button, the corresponding spreadsheet opens to be modified.

For more information regarding the equations that are used when either the fisher valve or relief valve is selected, refer to the section on **Choosing the Valve Equation**.

The flow rate spreadsheets are not always used. When a fisher valve or a relief valve is selected the standard unit operation is added to the subflowsheet.

Operation Modes

The operation modes available in the Dynamic Depressuring operation are as follows:

- Fire
- Fire Stephan Boltzman
- Fire API521
- Adiabatic

- Use Spreadsheet

The only time the values specified in a spreadsheet is not overwritten is when the Use Spreadsheet mode is selected.

You can view the results of the depressuring calculations in either tabular or graphical format.

The four types of depressuring calculations available are as follows:

Calculation	Description
Fire Mode	Used to simulate plant emergency conditions that 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. Do not specify a wetted area for this calculation.
Fire Stephan Boltzman	This mode includes radiation term, forced convection term, flame temperature, and ambient temperature term in the calculation.
Fire API521	The same as Fire Mode except the heat flux into the fluid is calculated from the API equations for a fire to a liquid containing vessel. A wetted area for the vessel is required, and is used for heat transfer in the model.
Adiabatic	Used to model the gas blow down of pressure vessels or piping. No external heat is applied. Heat flux between the vessel wall and the fluid is modeled as the fluid temperature drops due to the depressurization. The heat transfer coefficient is entered by the user, or can be calculated by HYSYS from the vessel fluid's vapour properties. When estimated by HYSYS, the heat transfer coefficient is estimated from the "wetted" area and the vessel volume specified by the user. The "wetted area" specified should be equal to the total surface area of the vessel, not the area in contact with the liquid. Typical use of this mode is the depressuring of compressor loops on emergency shutdown.
Use Spreadsheet	If you change the mode from **Use Spreadsheet** to another mode, the spreadsheet is over written.

Refer to **Dynamic Depressuring Subflowsheet** for more information on the **Use Spreadsheet** mode.

The Dynamic Depressuring utility can be used to simulate the depressuring of gas, gas-liquid filled vessels, pipelines, and systems with depressuring through a single valve. References to "vessel" can also be "piping" or "combinations of the two."

14.9.1 Design Tab

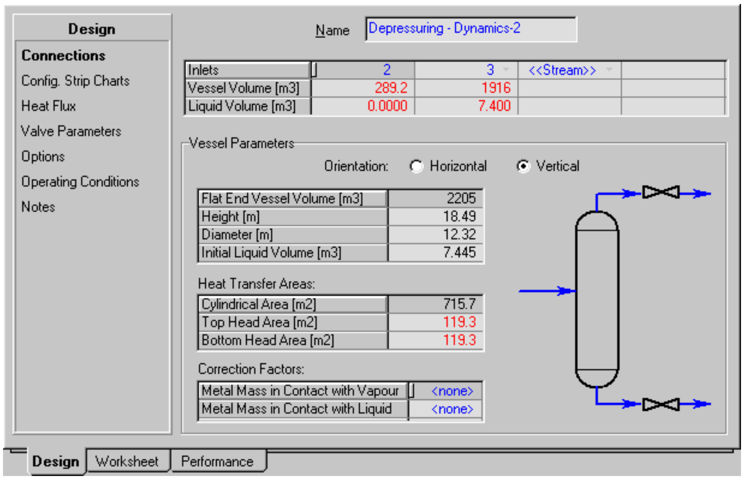
The Design tab contains the following pages:

- Connections
- Config. Strip Charts
- Heat Flux
- Valve Parameters
- Options
- Operating Conditions
- Notes

Connections Page

The Connections page allows you to specify the inlet stream, vessel volume, and initial liquid volume for the utility.

Figure 14.28



- **Name** field enables you to change the name of the dynamic depressuring utility.
- **Inlets** row enables you to enter or select up to four inlet streams for the utility.

Each stream has its own vessel volume and liquid volume.

- In the Vessel Parameters group, you can select the vessel's orientation using one of Orientation radio buttons (Horizontal or Vertical), and you can change the vessel surface area of the head for heat transfer calculations in the Heat Transfer Areas table.

  The parameters in the Correction Factors table are used to consider effects of the metal mass in contact with the liquid, and the metal mass in contact with the vapour.

If you enter only one inlet stream in the Vessel Parameters group, you must enter a volume for the vessel, liquid volume, and the height or diameter of the vessel; or enter the liquid volume, height, and diameter of the vessel. HYSYS then calculates the missing information.

If you enter more than one inlet stream, two rows appear underneath the Inlet row. The two rows are Vessel Volume and Liquid Volume. You can enter the vessel volume and the liquid volume for each stream in the associated Vessel Volume and Liquid Volume fields. Default values of the vessel parameters are calculated using a settle out calculation.

For more than one stream a settle out calculation is done. The results are approximate, because the settle out calculation is used in one vessel (like the Original Depressuring utility).

Larger systems and more complex configurations can be studied in Dynamics mode, where the pipe networks and so forth, can be configured.

The vessel/liquid volume fields are available for each inlet stream specified, and the fields are described in the section below.

Vessel\Liquid Volume Fields

For the stream selected for depressuring, HYSYS requires the vessel volume and the normal expected liquid volume of the vessel (in other words, at the normal liquid level). If the feed stream is two phase, the composition of the liquid is calculated from this.

You must either specify the Height and Diameter, or the Flat End Vessel Volume. If you specify only the Flat End Vessel Volume, HYSYS automatically estimates the Height, Diameter, and Initial Liquid Volume. By specifying only the vessel volume, the liquid volume is calculated using **Equation (14.5)** and the remainder of the vessel is assumed to be filled with equilibrium vapour.

$$Liquid\ Volume = Liquid\ Volume\ Flow\ of\ Liquid\ Phase \times 1\ hour \tag{14.5}$$

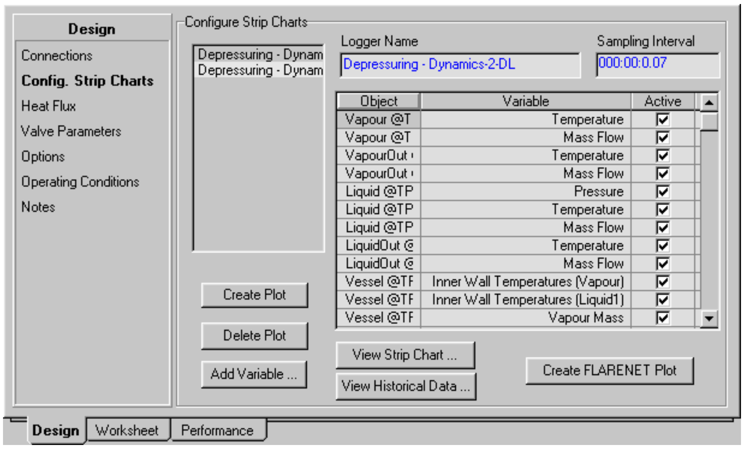
The Liquid Volume must be greater than 0 and less than the Flat End Vessel Volume.

If you specify both the Initial Liquid Volume and Flat End Vessel Volume, then the head space is assumed to be filled with equilibrium vapour.

Config. Strip Charts Page

The Config. Strip Charts page allows you to add, modify, or delete strip charts, and select the variables you want to appear in each strip chart.

Figure 14.29



You don't have to create a strip chart, because the dynamic depressuring utility automatically creates a *minimum required variable* strip chart.

When creating additional strip charts, ensure the select variables are from the correct depressuring subflowsheet.

The table below describes the objects available on the Config. Strip Charts page:

Object	Description
Logger Name field	The name of the selected strip chart on the list. You can change the name of the strip chart, by entering a new name in this field.
Sample Interval field	The length of time between data samples taken for the strip chart. The sample interval is set to equal the time step size specified on the Operating Conditions page of the Design tab. You have to run the Dynamic Depressuring utility calculations after the strip chart has been changed, or created in order to view the updated strip chart.
List	The list on the left side of the Logger Name field contains all the names of the strip charts associated with the current dynamic depressuring utility. You can manipulate the variables of strip charts by selecting the name of the strip chart on the list.
Table	Contains all the variables that can be stored in the strip chart. You can select, which variable you want to be stored in the strip chart during calculations by selecting the checkbox associated with the variable.
Create Plot button	Allows you to create a new strip chart. Click this button and a new strip chart appears in the list. The default name of the new strip chart is DataLogger.
Delete Plot button	Allows you to delete strip charts. Select the strip chart you want to delete from the list and click this button. This button is only available if there is a strip chart in the list.
Add Variable button	Allows you to enter a new variable in the strip chart. Select the strip chart you want the variable to appear in and click this button. The Variable Navigator property view appears, and you can select the variable you want to add from this property view.
View Strip Chart button	Allows you to view the strip chart. Select the strip chart you want to view from the list and click this button, the strip chart property view appears.

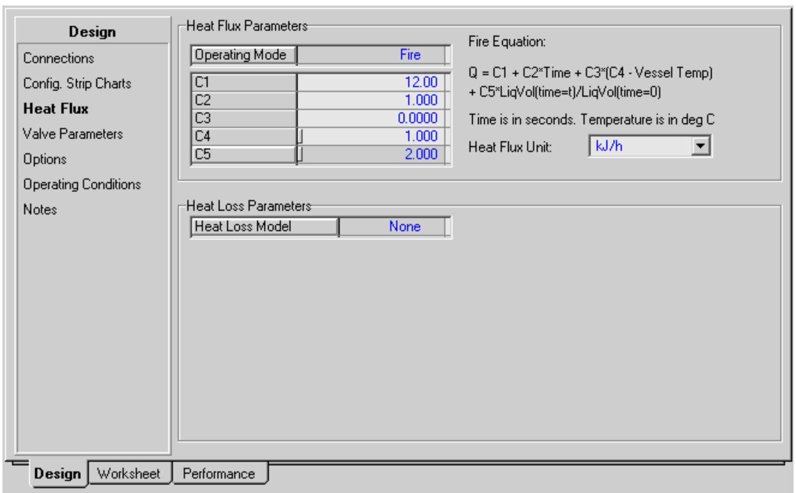
Refer to **Section 1.3.9 - Variable Navigator Property View** for information.

Object	Description
View Historical Data button	Allows you to view the data points stored in the strip chart in table format. Select the strip chart you want to view from the list and click this button, the Historical Data property view appears. The Historical Data property view contains two buttons that allow you to save/export the results: • Save To CSV File • Save To DMP File
Create FLARENET Plot button	Allows you to create a strip chart that contains information you may want to export to FLARENET. Click this button to create the strip chart. The default name is the dynamic depressuring utility's name followed by FLARENET. To add another FLARENET strip chart, change the default name of the previously created FLARENET strip chart before clicking the Create FLARENET Plot button again. You have to run the Dynamic Depressuring utility calculations after the strip chart has been changed, or created in order to view the updated strip chart.

Heat Flux Page

On the Heat Flux page, you can specify the depressuring mode and heat loss model for the utility.

Figure 14.30



Refer to **Section - Operation Modes** for a description of the available modes.

The available choices for depressuring modes are as follows:

- **Fire Mode**. When depressuring in Fire mode, five coefficients (C1 to C5) are required to set up the following generalized equation:

$$Q = C_1 + C_2 t + C_3(C_4 - T) + C_5\left(\frac{V_t}{V_0}\right) \tag{14.6}$$

where:

t = Time, seconds

T = Vessel temperature, °C

V_t *= Liquid volume at time = t*

V_o *= Liquid volume at time = 0*

As an example, you can model the standard heat transfer equation:

$$Q = UA\Delta T \tag{14.7}$$

By setting C_1, C_2, and C_5 to zero. Set C_3 to *UA* and C_4 to the constant temperature in the ΔT term.

- **Fire API521 Mode**. The Heat Flux Parameters page for depressuring in Fire API521mode is similar to the property view observed in Fire mode.

The wetted area is required for the Fire API521.

Three coefficients C1 to C3 need to be specified to set up the following equation, which is an extension to the standard API equation for flux to a liquid-containing vessel.

Depending on the version of HYSYS that you used to build the dynamic depressuring case, the heat flux is calculated differently.

For HYSYS 3.1 or older version:

$$Q = C_1 \cdot [\text{wetted area(time=t)}]^{C_2} \tag{14.8}$$

where:

$$wetted\ area(time{=}t) = wetted\ area(at\ time{=}0) \times \left\{1 - C_3\left[1 - \frac{LiqVol(time{=}t)}{LiqVol(time{=}0)}\right]\right\} \tag{14.9}$$

The units of the wetted area are controlled by the preferences, and not by the equation units.

For current version of HYSYS:

$$Q = C_1 \cdot [C_3 \cdot wetted\ area(time{=}t)]^{C_2} \tag{14.10}$$

Equation (14.10) uses a more rigorous method to calculate the wetted area by taking into account of the orientation of the depressuring vessel. In HYSYS 3.2 the depressuring utilities automatically use Equation (14.10) to calculate the heat flux. For cases that are built with HYSYS 3.1 or older version, when you re-run the depressuring utility in HYSYS 3.2 you have the option to calculate the heat flux using Equation (14.9) or Equation (14.10). However, once you selected Equation (14.10) and saved the case, you cannot re-calculate the heat flux using Equation (14.9).

- **Fire Stephan Boltzman Mode**.The following equation is used for the Fire Stephan Boltzman mode. This equation includes a radiation term, forced convection term, flame temperature term, and ambient temperature term.

$$Q = A_{total} \times (\varepsilon_f \times \varepsilon_v \times k \times (T_f + 273.15)^4 - (T_v + 273.15) + OutsideU \times (T_{ambient} - T_v)) \tag{14.11}$$

where:

A_{total} = *total surface area*

ε_f = *flame emissivity*

ε_v = *vessel emissivity*

k = *Boltzman constant*

T_f = *flame temperature*

T_v = *vessel temperature*

OutsideU = convective heat transfer coefficient between vessel and surrounding air

$T_{ambient}$ = ambient temperature

- **Adiabatic Mode**. When Adiabatic mode is selected, heat flux information is not required.
- **Use Spreadsheet Mode**. The Use Spreadsheet option refers to the duty spreadsheet of the Dynamic Depressuring utility used by the utility.

  This option allows you to edit the duty spreadsheet without the values in the spreadsheet getting overwritten when the utility runs. This option also allows the more advanced users the ability to use a different equation for the heat flux.

  When this option is selected, the **View Spreadsheet** button appears. Clicking the **View Spreadsheet** button opens the duty spreadsheet.

Heat Loss Parameters Group

The Heat Loss Model field contains a drop-down list, where you can select the heat loss model for the utility. There are three types of models:

- None
- Simple
- Detailed

Simple Model

The Simple model allows you to either specify the heat loss directly, or have the heat loss calculated from specified values:

- Overall U value
- Ambient Temperature

The heat transfer area, *A*, and the fluid temperature, T_f, are calculated by HYSYS. The heat loss is calculated using:

$$Q = UA(T_f - T_{amb}) \tag{14.12}$$

The simple heat loss parameters are:

- Overall Heat Transfer Coefficient
- Ambient Temperature

- Overall Heat Transfer Area
- Heat Flow

The Heat Flow is calculated as follows:

$$Heat\ Flow = UA(T_{Amb} - T) \tag{14.13}$$

where:

U = *overall heat transfer coefficient*

A = *heat transfer area*

T_{Amb} = *ambient temperature*

T = *holdup temperature*

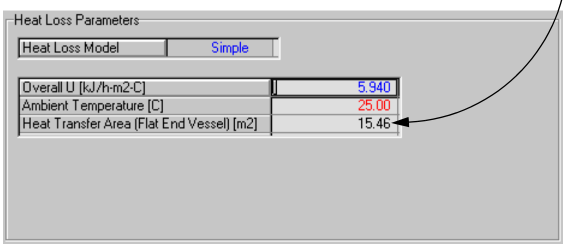
As shown in **Equation (14.13)**, Heat Flow is defined as the heat flowing into the vessel. The heat transfer area is calculated from the vessel geometry.

The overall heat transfer coefficient, U, and the ambient temperature, T_{Amb}, can be modified from their default values (shown in blue and red in the figure below).

Figure 14.31

The value in the Heat Transfer Area display field is based on the vessel geometry you entered on the Connections page.

Notice that the text is black, indicating the value in the field is calculated and cannot be changed on this page.



Detailed Model

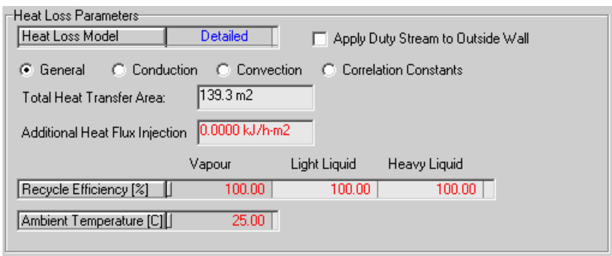
Refer to **Section 1.6 - HYSYS Dynamics** in the **HYSYS Dynamic Modeling** guide for more information.

The Detailed model allows you to specify more detailed heat transfer parameters.

You can either apply the duty to the fluid inside the vessel or to the external surface of the vessel. When you select the **Apply Duty Stream to Outside Wall** checkbox, HYSYS applies the duty calculations to the external surface of the vessel. When you clear the **Apply Duty Stream to Outside Wall** checkbox, HYSYS applies the duty calculations to the fluid inside the vessel.

When you select Detailed from the drop-down list, four radio buttons appear in the Heat Loss Parameters group.

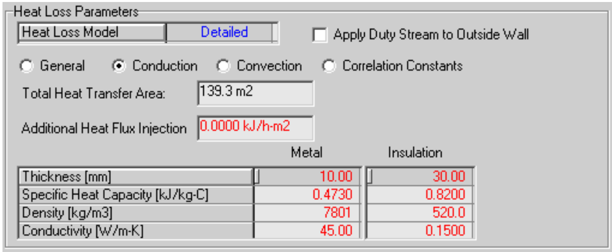
<table>
<tr><th>Radio Button</th><th>Description</th></tr>
<tr><td>General</td><td>Allows you to manipulate the Recycle efficiencies and Ambient temperature.



In the Additional Heat Flux Injection field you can specify a heat flux value for the fluid contents in the vessel.

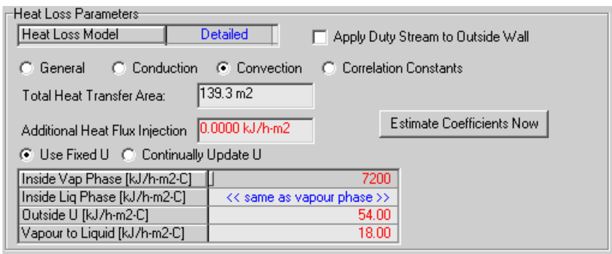
The Recycle efficiency default values are 100%, which means that all phases are always in thermodynamic equilibrium, and hence all phases have the same temperature.

If the efficiency values are reduced (to say 10%), then the vapor and liquid cannot reach equilibrium instantaneously and can have different temperatures. No single typical number can be suggested here, and you should try various scenarios for the possible temperature.</td></tr>
</table>

<table>
<tr><th>Radio Button</th><th>Description</th></tr>
<tr><td>Conduction</td><td>
Allows you to manipulate the conductive properties of the wall and insulation.


In the Additional Heat Flux Injection field you can specify a heat flux value for the fluid contents in the vessel.
You can specify the following properties:

Thickness of material. The insulation value thickness to be zero to model a vessel without insulation. The metal wall must have a finite thickness.
Specific Heat capacity of material
Density of material
Conductivity of material

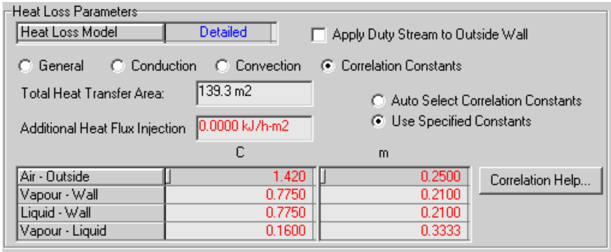
</td></tr>
<tr><td>Convection</td><td>
Allows you to manipulate the heat transfer coefficient for inside and outside the vessel, and between vapour and liquid material inside the vessel.


Fixed U. Select this radio button if you want the U values, that you specified, to be used throughout the calculations.
Update U. Select this radio button if you want the U values calculated using the current conditions during the calculations (in other words, no user input.). The U values are updated while solving.

Click the Estimate Coefficient button to estimate convective heat transfer coefficients (U value) using current conditions.
</td></tr>
</table>

<table>
<tr><th>Radio Button</th><th>Description</th></tr>
<tr><td>Correlation Constants</td><td>

Allows you to manipulate the heat transfer correlation coefficients.



- **Auto Select Correlation Constants**. Correlations constants are automatically selected based on the Grashof and Prandtl numbers.
- **Use Specified Constants**. You can specify the C and m constants in **Equation (14.14)** for the outside heat transfer coefficient for air.

The equation for outside heat transfer coefficient for air is:

$$h = C \times \left(\frac{\text{delta\_Temperature}}{\text{length}}\right)^m \tag{14.14}$$

All the other correlation are based on the equation below:

$$Nu = C \times (Gr \times Pr)^m \tag{14.15}$$

where:

Nu = Nusselt number

Gr = Grashof number

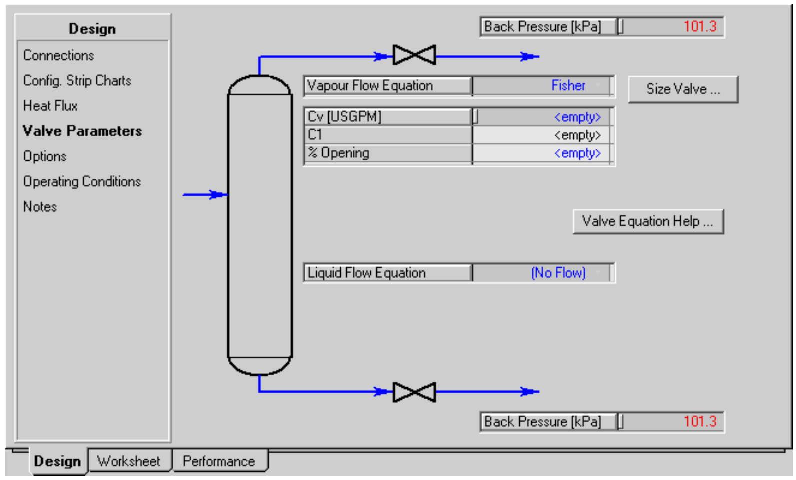
Pr = Prandtl number

</td></tr>
</table>

Valve Parameters Page

The Valve Parameters page allows you to select the type of valve equation you want to use for your vapour and liquid outlet streams.

Figure 14.32



Click the Valve Equation Help button to open the Depressuring Valve Equation Help property view, which contains a summary of all the valve equations available for the Dynamic Depressuring utility.

Choosing the Valve Equation

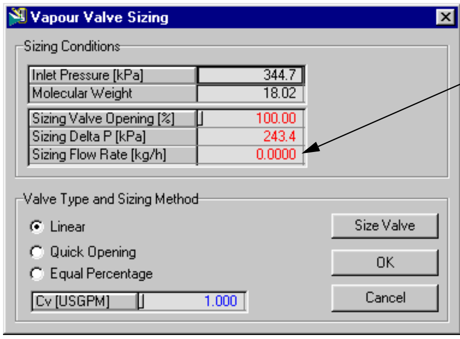
You can select the Valve Equation from the Vapour/Liquid Flow Equation drop-down list.

HYSYS recommends you use either the Fisher or Relief option to size the valve. The Fisher and Relief valve equations are more advanced than the other valve equations, and they can automatically handle choking conditions, and support various additional factors and options that can be accessed from the property views of the valve unit operation.

Use the Session Preferences property view to determine the units for the equation.

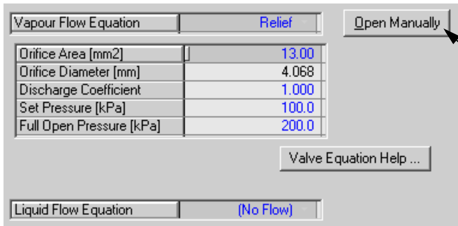
You have seven options for the valve equation, and they are listed in the following table:

Refer to **Section 7.6 - Valve** for more information about valves.

<table>
<tr><th>Equation</th><th>Description</th></tr>
<tr><td>Fisher</td><td>Uses a ‘Fisher’ valve (the standard valve in HYSYS). This option allows you to specify the Cv and % opening.
You can calculate Cv for a given flow rate:
1. Click the Size Valve button located next to the Vapour Flow Equation field. Notice that the Size Valve button only appears if you select the Fisher option. The Vapour/Liquid Valve Sizing property view appears.


When sizing the valve, you have to enter a Sizing Valve Opening (%).
2. Enter valve sizing conditions, and select the valve type and solving method.
3. Click the Size Valve button in the Valve Type and Sizing Method group, and notice a new Sizing Flow Rate will be calculated.
4. Click the OK button to accept the new size and exit the property view, or click the Cancel button to exit the property view without changing the valve size.</td></tr>
</table>

Refer to **Section 7.4 - Relief Valve** for more information about Relief valves.

Equation	Description
Relief valve	Uses the Relief valve operation in HYSYS. This option allows you to specify the orifice area, relief pressure, and full opening pressure.  You can open the Relief Valve manually by clicking the **Open Manually** button. You have to enter the following variables: • Orifice Area/Orifice Diameter • **Orifice Discharge Coefficient**. As fluid exits a reservoir through a small hole and enters another one, or flows out to the open air, stream lines tend to contract itself, mostly because of inertia. The coefficient of discharge is used to include this effect. If you do not want to include the effect, enter C = 1. • Relief Pressure • **Full Open Pressure**. To have the relief valve open all the time, set the full open pressure lower than the final expected vessel pressure, and set its set (or relief) pressure slightly lower than the full open pressure.
Supersonic	$$F = C_d A (P_1 \rho_1)^{0.5} \qquad (14.16)$$ *where:* *C_d = discharge coefficient (You can only enter values between 0 and 1 for the discharge coefficient.HYSYS recommends a value between 0.7 and 1 for the discharge coefficient.)* *A = area* *P_1 = upstream pressure* *ρ_1 = upstream density* Use the Supersonic equation for modeling systems when no detailed information is available on the valve. The flow through the valve is then proportional to *A* (area).

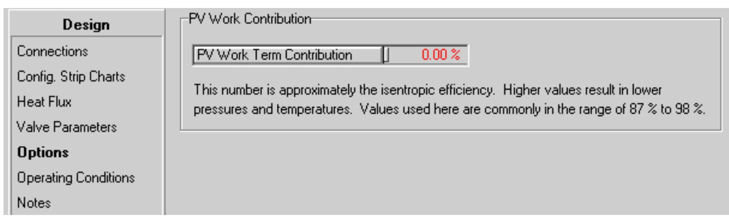
Equation	Description
Subsonic	$$F = C_d A\left[\frac{(P_1+P_{back})(P_1-P_{back})}{P_1}\rho_1\right]^{0.5} \quad (14.17)$$ *where:* C_d = *discharge coefficient (You can only enter values between 0 and 1 for the discharge coefficient. HYSYS recommends a value between 0.7 and 1 for the discharge coefficient.)* A = *area* P_1 = *upstream pressure* P_{back} = *back pressure or valve outlet pressure* ρ_1 = *upstream density* If the pressure in the vessel is such that there is sub-critical flow (generally upstream pressure less than twice the backpressure), then you have no option but to use the Subsonic Equation. This equation is used in the same instances as the Supersonic equation except when you have subsonic flow. By applying this equation, you are required to specify P_{back} or the valve back pressure. By specifying P_{back} to be slightly less than the Relief Pressure, it is possible to have your depressuring analysis cycle between pressure build-up and relief. Ensure a reasonable pressure differential, and increase the number of pressure steps for the analysis.
Masoneilan	$$F = C_1 C_v C_f Y_f (P_1 \rho_1)^{0.5} \quad (14.18)$$ *where:* C_1 = *1.6663 (SI default) or 38.86 (Field default) You cannot change the value for* C_1. C_v = *valve coefficient* C_f = *critical flow factor* P_1 = *upstream pressure* ρ_1 = *upstream density* $Y_f = y - 0.148 y^3$ *(the max value of* Y_f *is 1)* y = *expansion factor* Taken from the Masoneilan catalogue, this equation can be used for general depressuring valves to flare. Often the C_v for a valve is known from vendor data so when Masoneilan is selected, the appropriate values for C_1 and C_2, are automatically set as well as the units.

Equation	Description
General	$$F = C_d A_v K_{term}(g_c P_1 \rho_1 k)^{0.5} \qquad (14.19)$$ *where:* C_d = *discharge coefficient (You can only enter values between 0 and 1 for the discharge coefficient. HYSYS recommends a value between 0.7 and 1 for the discharge coefficient.)* A_v = *valve orifice area* g_c = *dimensionless constant = 1.0* $kg.m/N.s^2$ *(32.17* $lb.ft/lb_f.s^2$*)* k = *ratio of specific heats (Cp/Cv)* P_1 = *upstream pressure* ρ_1 = *upstream density* This equation is from *Perry's Chemical Engineering Handbook*. Refer to it if you know the valve throat area. This equation makes certain limiting assumptions concerning the characteristics of the orifice.
No Flow	Indicates that there is no flow rate output in the valve.
Use Spreadsheet	The flow rate spreadsheet in the Dynamic Depressuring subflowsheet used by the utility. This option allows you to edit the spreadsheet without the changes made in the spreadsheet getting overwritten when the utility runs. This option also allows more advanced users to define a completely different equation for the flow.

Options Page

The Options page allows you to enter a value for the PV Work Term Contribution.

Figure 14.33



The PV Work Term Contribution value is used to approximate the isentropic efficiency. Higher values result in lower pressures and temperatures, and the commonly used values range from 87% to 98%.

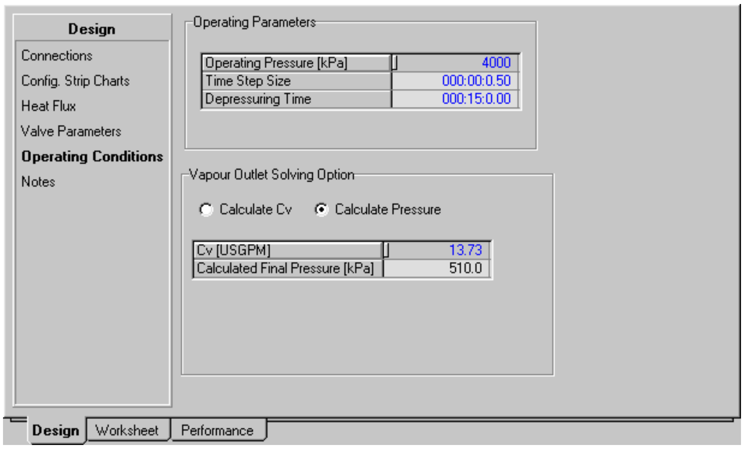
Operating Conditions Page

On the Operating Conditions page, you can specify what you want to solve (valve coefficient or pressure).

HYSYS can solve either the valve coefficient or the pressure.

The information required on this page varies, depending on the type of valve equation you selected for the vapour flow on the Valve Parameters page.

Figure 14.34



The three specifications that apply to all valve equations are:

Specification	Description
Operating Pressure	Allows you to enter the initial vessel pressure. The default value is set to the pressure of the inlet stream. When the utility has multiple inlet streams, then only the operating pressure is calculated (settle out calculations).
Time Step Size	Allows you to specify the integration step size. The default value is 0.5 seconds. Helpful Tip: reduce the time step when you see a large flow rate compared to the volume.
Depressuring Time	The Depressuring Time is the time you want this operation to take. It is defaulted as 15 minutes (900 seconds) based on API 521, but you can alter this if required.

The Dynamics Depressuring utility runs the dynamics integrator. The integrator uses a fixed integration step size with a default value of 0.5 seconds, and always runs for the total depressuring time specified. If your vessel depressurizes in relatively short time (for example 3 seconds), you may need to decrease the integration step size and depressuring time appropriately. The sampling frequency of the strip chart is automatically set to the value of the Time Step Size.

The table below describes the required specifications for each equation when you select the Calculate CV or Calculate Pressure radio button.

Valve Equation	Required Specification(s)	
	Calculate CV	**Calculate Pressure**
Fisher / Masonelian	• Initial C_v Estimate • Max C_v Step Size • Pressure Tolerance • Maximum number of Iterations • Iteration Count • **Final Pressure**. Based on API, it is normal to depressure to 50% of the starting pressure or to 100 psig (6.89 barg). If the depressuring time is reached (for API 521, 15 minutes) before the final pressure is achieved, then calculations stop. The final pressure is used to calculate C_v for the vapour outlet stream.	• C_v

Valve Equation	Required Specification(s)	
	Calculate CV	**Calculate Pressure**
Relief	• Initial Orifice Area Estimate • Max Area Step Size • Pressure Tolerance • Maximum number of Iterations • Iteration Count • **Final Pressure**. Based on API, it is normal to depressure to 50% of the starting pressure or to 100 psig (6.89 barg). If the depressuring time is reached (for API 521, 15 minutes) before the final pressure is achieved, then calculations stop. The final pressure is used to calculate the orifice area for the vapour outlet stream.	• Orifice Area
Supersonic / Subsonic / General	• Initial Area Estimate • Max Area Step Size • Pressure Tolerance • Maximum number of Iterations • Iteration Count • **Final Pressure**. Based on API, it is normal to depressure to 50% of the starting pressure or to 100 psig (6.89 barg). If the depressuring time is reached (for API 521, 15 minutes) before the final pressure is achieved, then calculations stop. The final pressure is used to calculate area for the vapour outlet stream.	• Area

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor, where you can record any comments or information regarding the utility, or to your simulation case in general.

14.9.2 Worksheet Tab

Refer to the **Section 1.3.10 - Worksheet Tab** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

14.9.3 Performance Tab

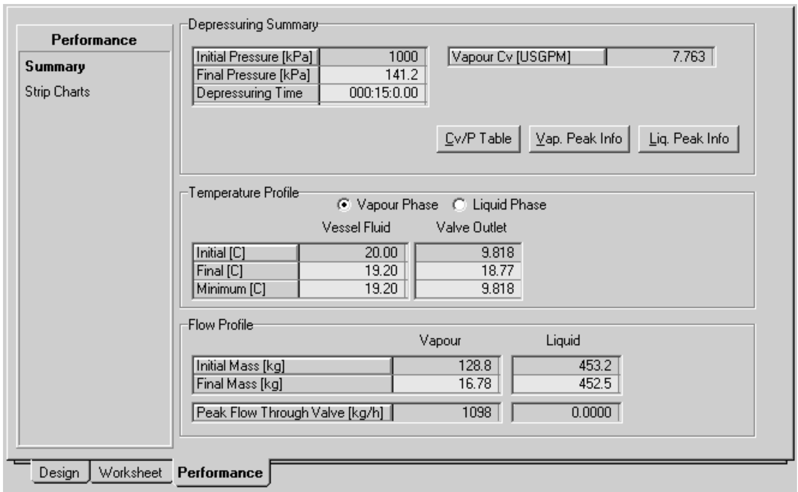
The Performance tab contains the following pages:

- Summary
- Strip Charts

Summary Page

The Summary page contains a summary of all the calculated results.

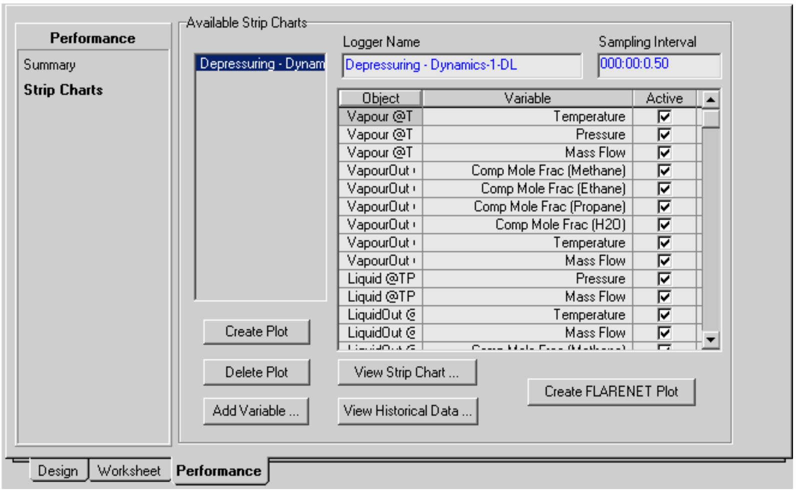
Figure 14.35



Strip Charts Page

The Strip Charts page allows you to view the results in tabular or graphical format.

Figure 14.36



The page contains five buttons:

Refer to **Section 1.3.9 - Variable Navigator Property View** for information.

- **Create Plot**. Generates a strip chart.
- **Delete Plot**. Allows you to delete a strip chart.
- **Add Variable**. Allows you to add the strip chart variables.
- **View Strip Chart**. Allows you to open and view the strip chart.
- **View Historical Data**. Allows you to open and view the historical data. The Historical Data property view contains two buttons that allow you to save/export the results: Save To CSV File and Save To DMP File.
- **Create FLARENET Plot**. Allows you to generate a FLARENET strip chart.

The Strip Charts page allows you easy access to the strip charts and the historical data.

14.10 Envelope Utility

To add the Envelope utility, refer to the section on **Adding a Utility**.

The Envelope utility allows you to examine relationships between selected parameters for any two-phase or three-phase stream of known composition, including streams with only one component.

The Envelope utility is restricted to the Peng Robinson and Soave Redlich Kwong equations of state.

14.10.1 HYSYS Two-Phase Envelope

The Vapour-Liquid Envelopes can be plotted for the following variables:

- Pressure-Temperature
- Pressure-Volume
- Pressure-Enthalpy
- Pressure-Entropy
- Temperature-Volume
- Temperature-Enthalpy
- Temperature-Entropy

For the Pressure-Temperature envelope, quality lines, and a hydrate curve can also be added to the plot. The remaining curves allow the inclusion of Isocurves (Isotherms or Isobars).

Since the Envelope is calculated on a dry basis, you must be careful when applying the utility to multi-component mixtures that contain H_2O or any other component which can form a second liquid phase.

Design Tab

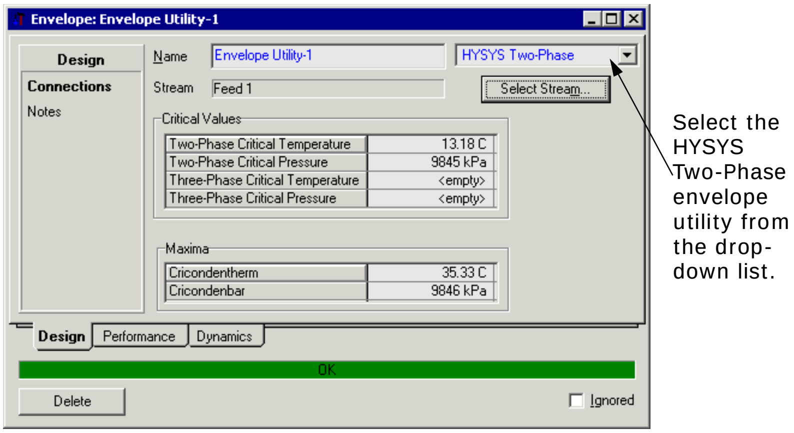
The Design tab contains the following pages:

- Connections
- Notes

Connections Page

On the Connections page, you can select the HYSYS Two-Phase envelope utility and attach it to a stream. The Connections page also displays, the calculated critical temperature and pressure, as well as the cricondentherm and cricondenbar.

Figure 14.37



You can change the name of the envelope utility by typing in the Name field. The stream is chosen from the Select Process Stream property view, which is accessed by clicking the Select Stream button.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page/Tab**.

The Notes page provides a text editor, where you can record any comments or information regarding the utility, or to your simulation case in general.

Performance Tab

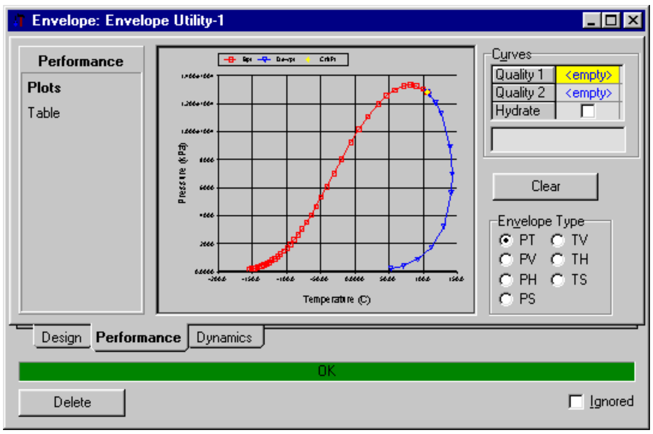
The Performance tab contains the following pages:

- Plots
- Table

Plots Page

On the Plots page, you can display different types of envelope graphs depending on the selected radio button in the Envelope Type group.

Figure 14.38



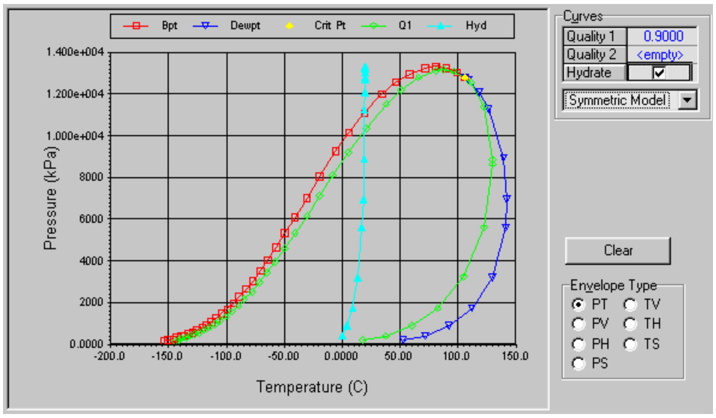
You can clear all curves on the plot at any time by clicking the Clear button.

The following sections discuss the various available envelopes in more detail.

Pressure-Temperature Envelope

When you select the PT radio button in the Envelope Type group, the Vapour-Liquid Envelope for a quality of 1.0 automatically appears.

Figure 14.39



The plot on the right, shows an envelope for a quality of 0.9. A quality of 0.9 is represented by two curves; one with a vapour fraction of 0.9 and the other having a liquid fraction of 0.9.

This is actually represented by two curves; one with a vapour fraction of 1.0 and the other having a liquid fraction of 1.0. These curves meet at the stream critical point. You can plot additional envelopes for different qualities simply by typing the desired quality (between 0 and 1) in the Quality 1 and Quality 2 fields.

For more information on the Hydrate calculation models, refer to **Section 14.12 - Hydrate Formation Utility**.

Select the **Hydrate** checkbox to have HYSYS calculate and display the hydrate temperature curve for pressures up to the cricondenbar. When you select the **Hydrate** checkbox, you can select from the drop-down list the model (Assume Free Water, Asymmetric, Symmetric or Vapour Phase Only) to perform the Hydrate Formation calculations.

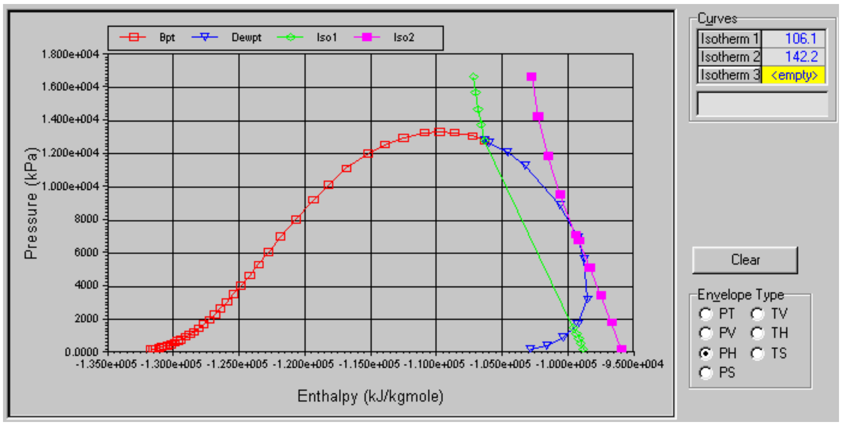
PV-PH-PS Envelopes

If you select the PV radio button, the Pressure-Volume Envelope appears. If you select the PH radio button, the Pressure-Enthalpy Envelope appears. If you select the PS radio button, the Pressure-Entropy Envelope appears.

For each of these Envelopes, you can display a maximum of three Isotherms (constant temperature curves) by entering values in the Curves group.

The figure below shows the Pressure-Enthalpy Envelope for a stream, with 106°C and 142°C Isotherms.

Figure 14.40



TV-TH-TS Envelopes

If you select the TV radio button, the Temperature-Volume Envelope appears. If you select the TH radio button, the Temperature-Enthalpy Envelope appears. If you select the TS radio button, the Temperature-Entropy Envelope appears.

For each of these Envelopes, you can display up to three Isobars (constant pressure curves). Simply, enter the desired pressure(s) in the Curves group.

The figure below shows the Temperature-Entropy envelope for a stream, with a 2068 kPa Isobar.

Figure 14.41

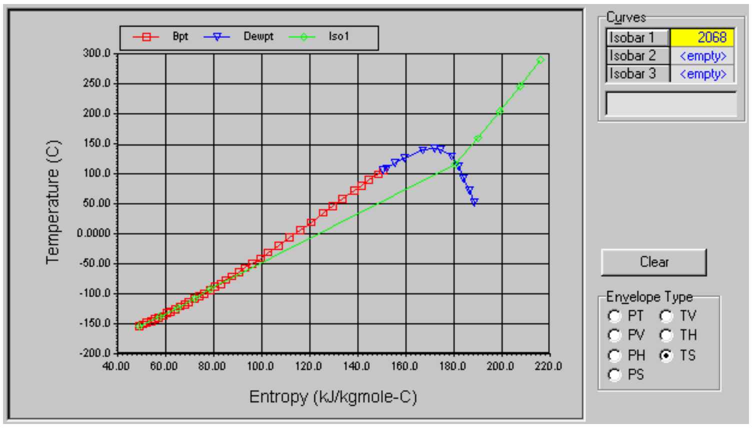


Table Page

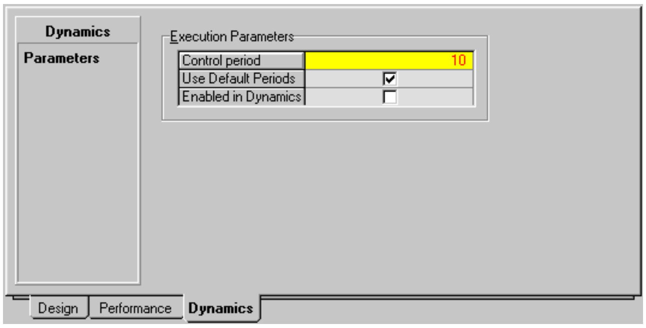
You can view the envelope results in tabular format on the Table page. To view the tabular results of different envelopes, from the Table Type drop-down list select the table type for the data. All Isocurves and Quality lines associated with the individual envelopes are transferred to the table.

Just like the Clear button in the Plots page, you can clear all curves data at any time by clicking the Clear button in the Table page.

Dynamics Tab

The Dynamics tab allows you to control how often the utility gets calculated when running in Dynamic mode.

Figure 14.42



The Control Period field is used to specify the frequency that the utility is calculated. A value of 10 indicates that the utility be recalculated every 10th pressure flow step. This can help speed up your dynamic simulation since utilities can require some time to calculate.

The **Use Default Periods** checkbox allows you to set the control period of one utility to equal the control period of any other utilities that you have in the simulation. For example, if you have five utilities and require them all to have a control period of 5 and currently the value is 8, with this checkbox selected if you change the value in one utility all the other utilities change. Alternatively, if you want all the utilities to have different values you would clear this checkbox.

The **Enable in Dynamics** checkbox is used to activate this feature for use in Dynamic mode.

14.10.2 Three-phase Envelope Utility

The three-phase envelope utility allows you to examine the relationships between selected parameters, for any stream of known composition, including streams with components that can potentially form a second liquid phase (for example, water, methanol, H2S, and so forth). Vapour-Liquid, Liquid-Liquid, and Vapour-Liquid-Liquid envelopes can be plotted for the following variables:

- Pressure-Temperature
- Pressure-Volume
- Pressure-Enthalpy
- Pressure-Entropy
- Temperature-Volume
- Temperature-Enthalpy
- Temperature-Entropy

The three-phase envelope is designed for the use in the oil and gas industries, which deal primarily with water, alcohols, hydrocarbon and sour gas components, and use an equation-of-state to model their fluid systems. The three-phase envelope utility can be used for a wide variety of systems and property packages.

It is recommended that you ignore the three-phase envelope utility during calculation, as the utility may be slow in calculating phase envelopes of streams containing a large number of components.

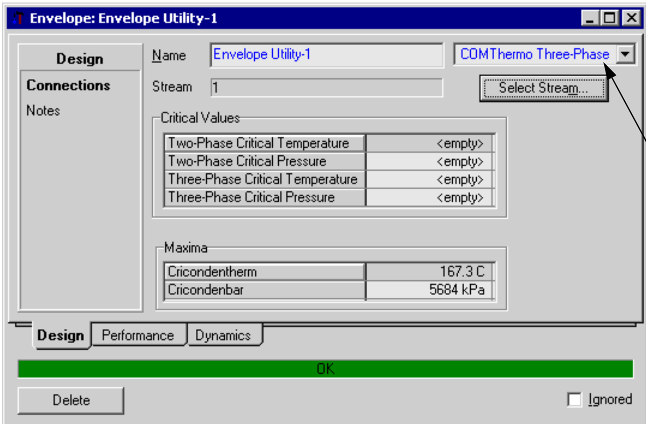
You can select the Ignored checkbox on the utility's property view to ignore this utility during calculations.

Design Tab

The Design tab contains the following pages:

- Connections
- Notes

Figure 14.43



Select the COMThermo Three-Phase utility from the drop-down list.

Refer to **Section 14.2.1 - Design Tab** for more information on the Connections and Notes page.

You can select COMThermo Three-Phase from the drop-down list.

Performance Tab

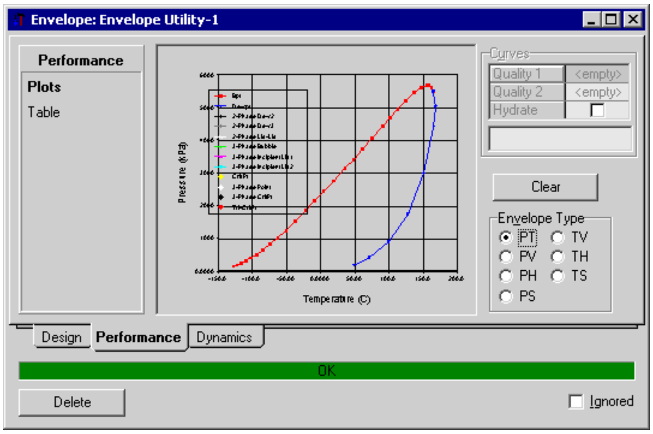
The Performance tab contains the following pages:

- Plots
- Table

Plots Page

The Plots page allows you to display different types of envelope graphs depending on the selected radio button in the Envelope Type group.

Figure 14.44



The following types of points are supported in the three-phase envelope utility:

Type of Point	Description
Two-phase Dew Line 1	Liquid 1 incipient
Two-phase Dew Line 2	Liquid 2 incipient
Two-phase Bubble Line	Vapour incipient
Two-phase Liquid-Liquid Line	Liquid 1 or Liquid 2 incipient
Two-phase Critical Point	Vapour/Liquid critical on two-phase line
Two-phase Cricondentherm	Maximum Temperature Point
Two-phase Cricondenbar	Maximum Pressure Point
Three-phase Point	Vapour/Liquid or Liquid/Liquid incipient
Three-phase Critical Point	Vapour/Liquid or Liquid/Liquid incipient
Three-phase Tri-Critical Point	Vapour, Liquid 1 & Liquid 2 all critical
Three-phase Bubble Line	Vapour incipient
Three-phase Incipient Liquid Line 1	Liquid 1 incipient
Three-phase Incipient Liquid Line 2	Liquid 2 incipient

The following sections discuss the various available envelopes in more detail.

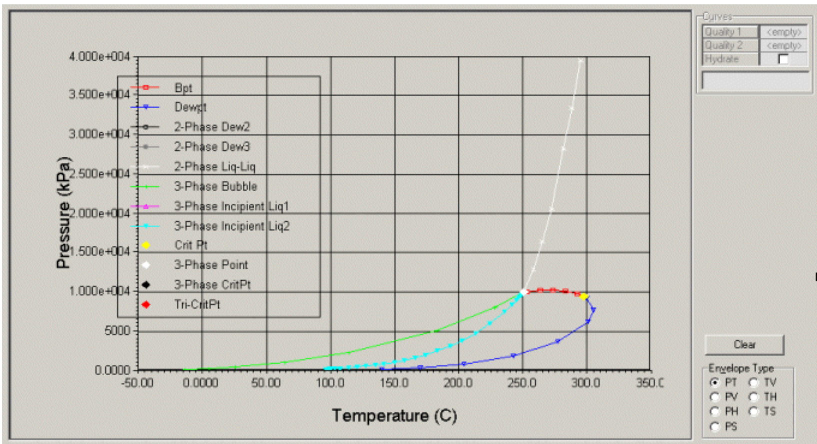
Pressure-Temperature Envelope

When you select the PT radio button in the Envelope Type group, the Vapour-Liquid, Liquid-Liquid or Vapour-Liquid-Liquid envelopes automatically appears. The following are the various scenarios possible:

Instability on the Two-phase Bubble Line

This scenario results in envelopes that exhibit a two-phase dew line and a two-phase bubble line that intersect at a critical point. A three-phase point is found on the two-phase bubble line from which emerge three branches. These include the three-phase bubble line where the vapour phase is incipient, a three-phase incipient liquid line where the liquid phase is incipient and the two-phase liquid-liquid line where one of the liquid phases is incipient. This is common when a second liquid forming component such as H_2O is present along with some heavy hydrocarbons (>C7). An equimolar mixture of o-xylene, p-xylene, H_2S and H_2O exhibit this behavior as shown below:

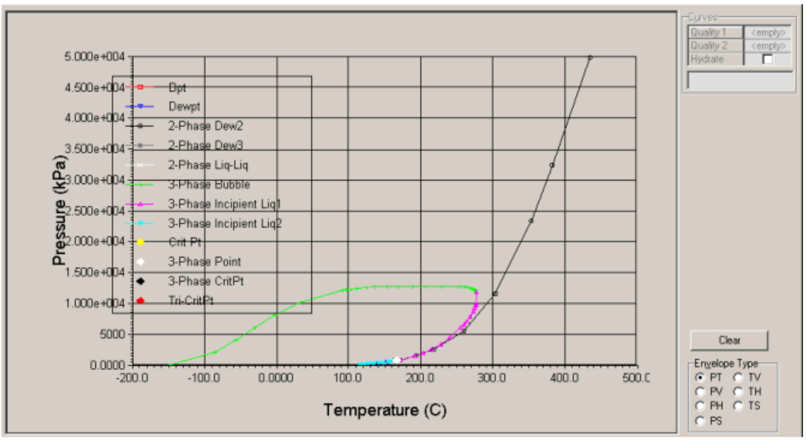
Figure 14.45



Instability on the Two-phase Dew Line

This scenario results in envelopes, which exhibit two, two-phase dew lines which intersect at a three-phase point. From the three-phase point emerge two three-phase branches where each of the liquids is incipient. There is also a three-phase bubble line that intersects one of the incipient liquid lines at the three-phase critical point. An example of this scenario is shown in a methane, n-decane and water mixture shown below:

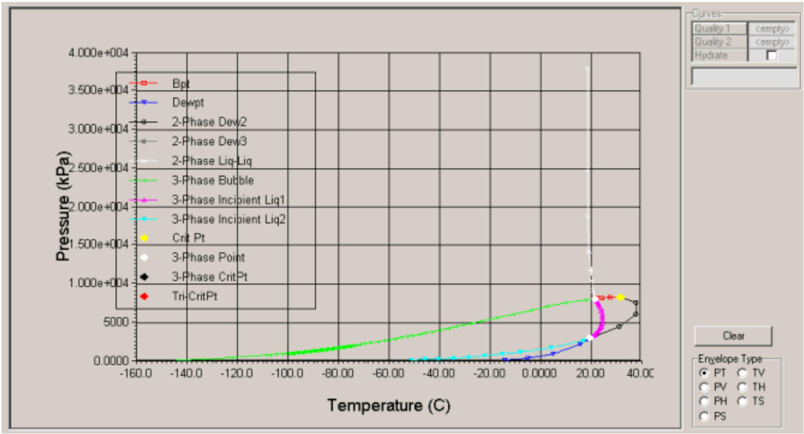
Figure 14.46



Instability on both the Two-phase Dew and the Two-phase Bubble Lines

This scenario results in envelopes, which exhibit two, two-phase dew lines which intersect at a three-phase point. From the three-phase point emerge two three-phase branches where each of the liquids is incipient. It also exhibits a two-phase dew line and a two-phase bubble line that intersect at a critical point. A three-phase point is found on the two-phase bubble line from which emerge three branches. These include the three-phase bubble line where the vapour phase is incipient, a three-phase incipient liquid line where the liquid phase is incipient and the two-phase liquid-liquid line where one of the liquid phases is incipient. This is common when the amount of second liquid forming component such as water is present in small amounts along with hydrocarbon components. An equimolar mixture of C1, C3 and CO2 with a small amount of H2O exhibit this behavior as shown below:

Figure 14.47

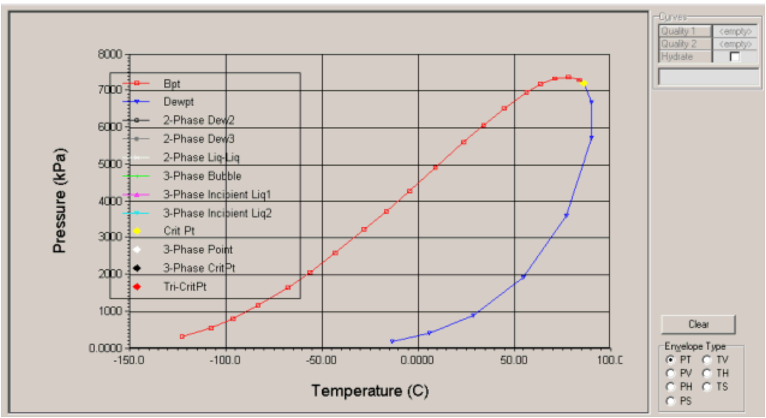


Non-Instability on the Two-phase Dew or the Two-phase Bubble Lines

This results is several sub-scenarios as follows:

- **No three-phase region present**. This scenario exhibits the regular two-phase envelope with no three-phase region. The two-phase bubble and two-phase dew lines intersect at a critical point. This is common where there is no second liquid forming components in the system. An equimolar mixture of C1, C2, C3, and n-C4 exhibit this behavior as show below:

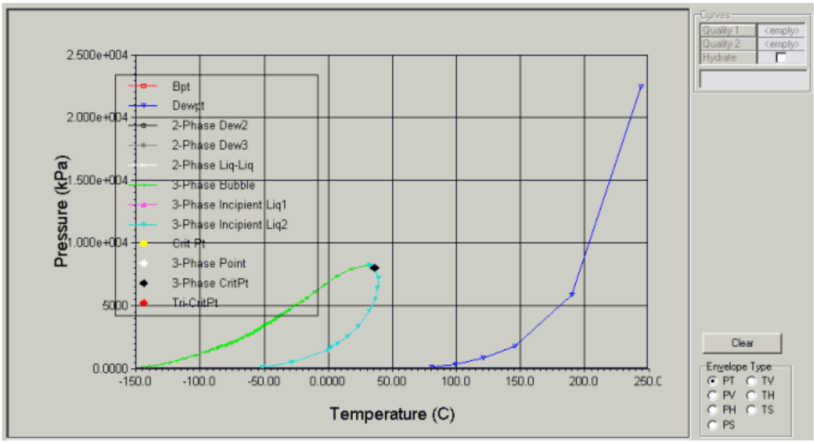
Figure 14.48



- **Three-phase region present along with three-phase critical point but no two-phase critical point**. This scenario exhibits a two-phase dew line without a critical point found on it. A three-phase region is present that is bounded by a three-phase bubble line and a three-phase incipient liquid line. The incipient vapour and incipient liquid are critical at the three-phase critical point. This is common when a second liquid forming component such as H_2O is present along with light hydrocarbons (<C7).

An equimolar mixture of C1, C3, CO2, and H2O exhibit this behavior as shown below:

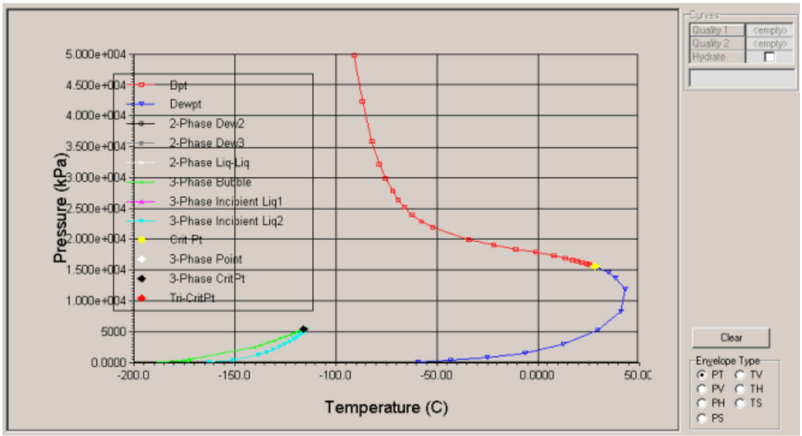
Figure 14.49



- **Three-phase region present along with a two-phase critical point and a three-phase critical point**. This scenario exhibits a two-phase dew line and a two-phase bubble line that intersect at a critical point. A three-phase region is present that is bounded by a three-phase bubble line and a three-phase incipient liquid line. The incipient vapour and incipient liquid are critical at the three-phase critical point.

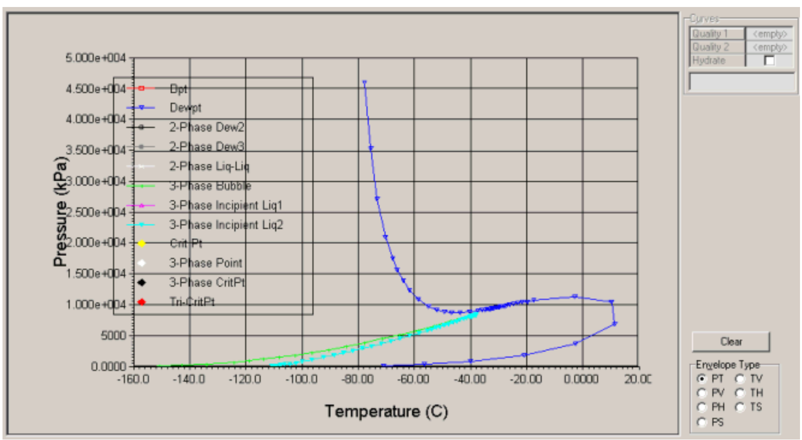
An equimolar mixture of C1, COS, CO2 and H2S and N2 exhibit this behavior as shown below:

Figure 14.50



- **Three-phase region present with two-phase critical point or three-phase critical point not present**. This scenario exhibits a two-phase dew line. The three-phase region present is bounded by a three-phase bubble line and a three-phase incipient liquid line. The two three-phase lines do not intersect at a critical point. A mixture of C1, C2, C3, CO2 and H2S exhibits this behavior as shown below:

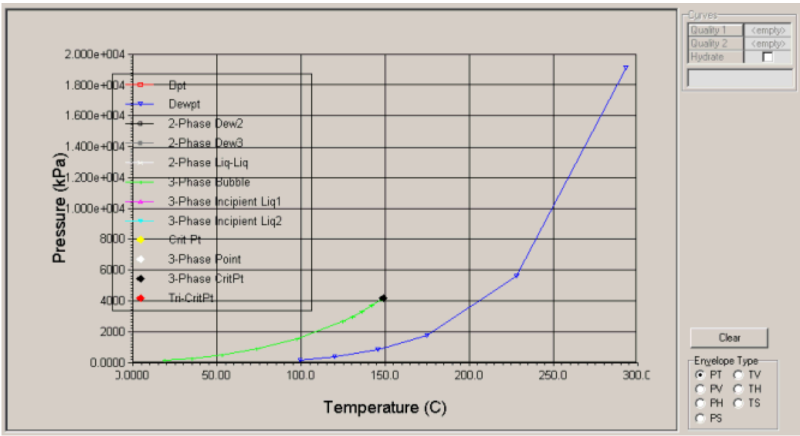
Figure 14.51



The Three-phase envelope utility can also plot three phase envelopes for binary mixtures containing a second liquid forming component. Typically a three-phase critical point is observed for such systems. There are two scenarios with this case:

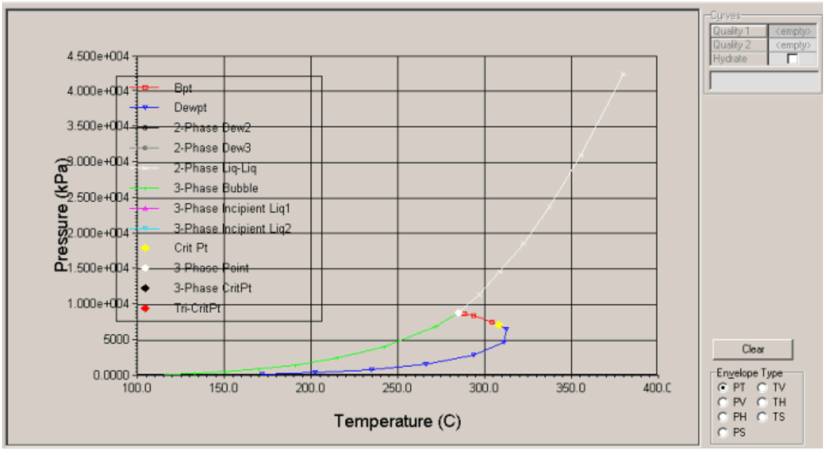
- The second liquid forming component is the light component. An equimolar mixture of water and n-butane is an example of such an envelope as shown below:

Figure 14.52



- The second liquid forming component is the heavy component. An equimolar mixture of water and n-decane is an example of such an envelope as shown below:

Figure 14.53

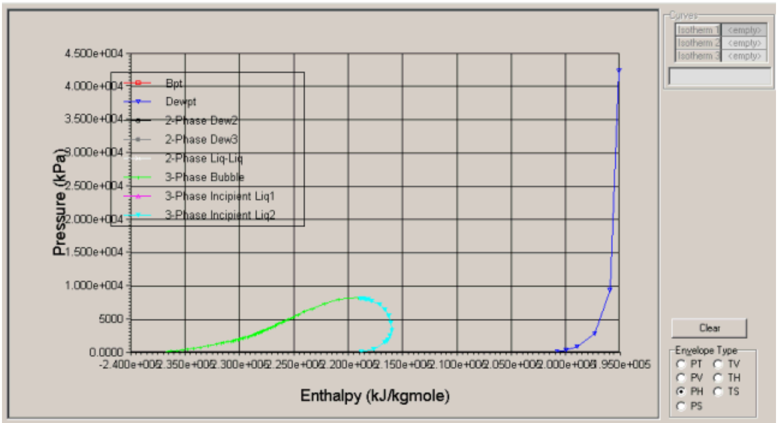


The hydrate formation curve and the quality lines are not available with the Three-phase envelope utility. You can clear all curves (except the default) at any time by clicking the Clear button.

PV-PH-PS Envelopes

If you select the PV radio button, the Pressure-Volume Envelope appears. If you select the PH radio button, the Pressure-Enthalpy Envelope appears. If you select the PS radio button, the Pressure-Entropy Envelope appears. The figure below shows the Pressure-Enthalpy Envelope for a stream:

Figure 14.54



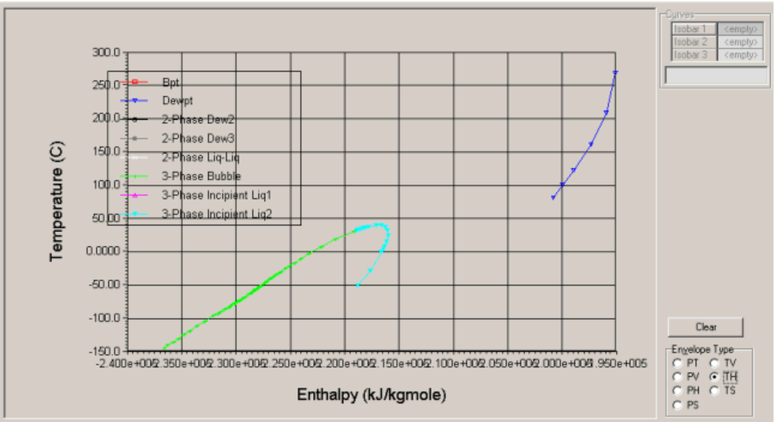
Isotherms are not available with the Three-phase envelope utility.

You can clear all curves (except the default) at any time by clicking the Clear button.

TV-TH-TS Envelopes

If you select the TV radio button, the Temperature-Volume Envelope appears. If you select the TH radio button, the Temperature-Enthalpy Envelope appears. If you select the TS radio button, the Temperature-Entropy Envelope appears. The figure below shows the Temperature-Entropy Envelope for a stream.

Figure 14.55

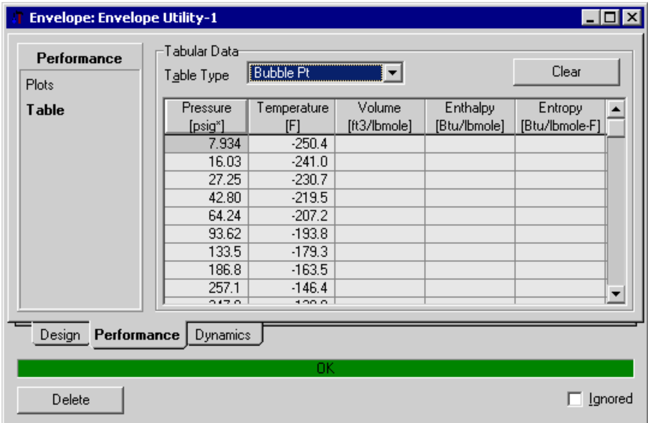


Isotherms are not available with the Three-phase envelope utility.

Table Page

You can view the envelope results in tabular format on the Table page. To view the tabular results of different envelopes, from the Table Type drop-down list select the table type for the data.

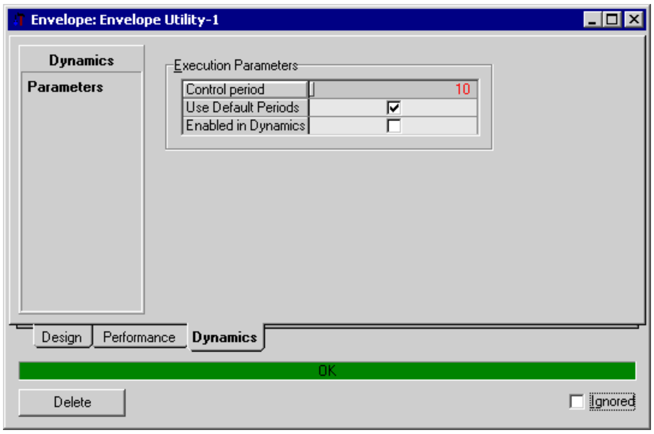
Figure 14.56



Dynamics Tab

The Dynamics tab allows you to control how often the utility gets calculated when running in Dynamics mode.

Figure 14.57



The Control Period field is used to specify the frequency that the utility is calculated. A value of 10 indicates that the utility will be recalculated every 10th pressure flow step. This can help speed up your dynamic simulation since utilities can require some time to calculate.

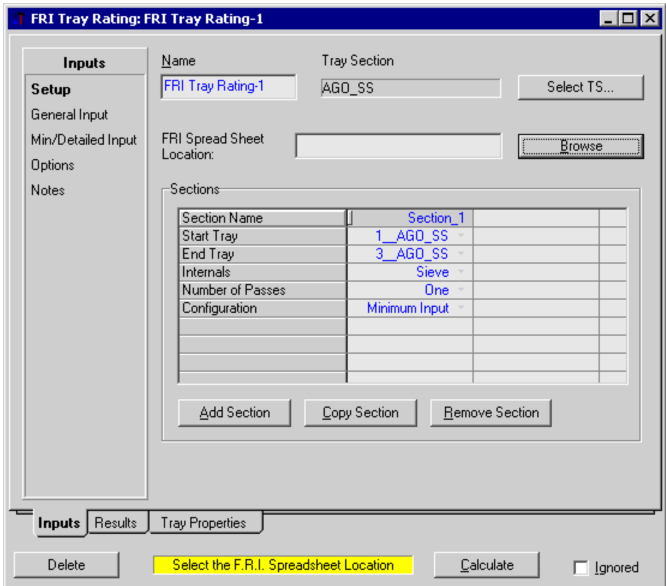
The **Use Default Periods** checkbox allows you to set the control period of one utility to equal the control period of any other utilities that you have in the simulation. For example, if you have five utilities and require them all to have a control period of 5 and currently the value is 8, with this checkbox selected if you change the value in one utility all the other utilities change. Alternatively, if you want all the utilities to have different values you would clear this checkbox.

The **Enable in Dynamics** checkbox is used to activate this feature for use in Dynamics mode.

14.11 FRI Tray Rating Utility

The FRI Tray Rating utility enables you to use the seive tray spreadsheet from FRI (Fractionation Research Institute) to size or rate tray sections from a solved HYSYS simulation case.

Figure 14.58



You must have the FRI spreadsheet in order to use this utility. Refer to www.fri.org for more information.

To begin FRI calculation, ensure all the required data has been entered (see the message in the status bar) and click the **Calculate** button.

14.11.1 Inputs Tab

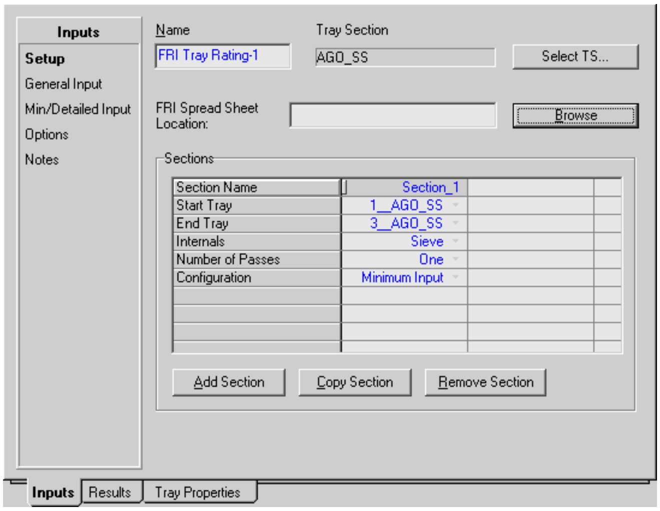
The Inputs tab contains the options to setup the FRI Tray Rating, the options are grouped in the following pages:

- Setup
- General Input
- Min/Detailed Input
- Options
- Notes

Setup Page

The Setup page enables you to select the tray section to be analysed, select the FRI spread sheet file, modify the name of the utility, and configure or modify the selected tray section.

Figure 14.59



Object	Description
Name field	Enables you to modify the name of the FRI Tray Rating utility.
Tray Section field	Displays the selected column for analysis.
Select TS button	Enables you to select the column you want to analyse.

Object	Description
FRI Spread Sheet Location field	Displays the location of the selected FRI spread sheet file.
Browse button	Enables you to find and select the FRI spread sheet file. The FRI spread sheet file is an Excel file (*.xls).
Sections table	Enables you to configure the following properties of the tray section: • the name of the tray section • the location of the first/starting tray • the location of the last/end tray • the internal tray type • the number of passes • the configuration type: Minimum Input or Detailed Input (tray section requires two or more passes before the Detailed Input option is available)
Add Section button	Enables you to add more sections to the selected column.
Copy Section button	Enables you to add a copy of the selected existing section in the Section table.
Remove Section button	Enables you to remove the selected section in the Section table.

To select a tray section from an existing column in the simulation:

1. Click the **Select TS** button. The Select Tray Section property view appears.
2. In the Flowsheet list, select the flowsheet containing the tray section you want.
3. In the Object list, select the column you want to analyse.
4. Click the **OK** button. The selected column will appear in the **Tray Section** display field.

To select the FRI spread sheet file:

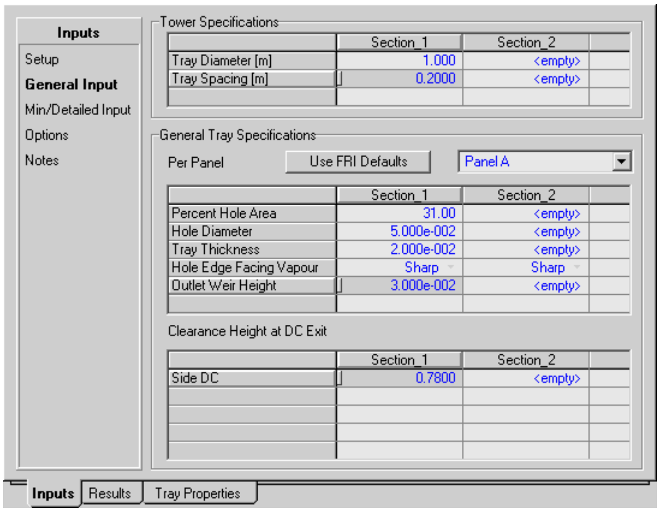
1. Click the **Browse** button. The Select the location of the desired FRI spreadsheet property view appears.
2. In the **Look in** drop-down list, locate the drive and folder containing the FRI spread sheet file.
3. Select the FRI spread sheet file name and click the **Open** button.

   The path and name of the selected FRI spread sheet file appears in the **FRI Spread Sheet Location** display field.

General Input Page

The General Input page enables you to specify general information about the selected tray section.

Figure 14.60



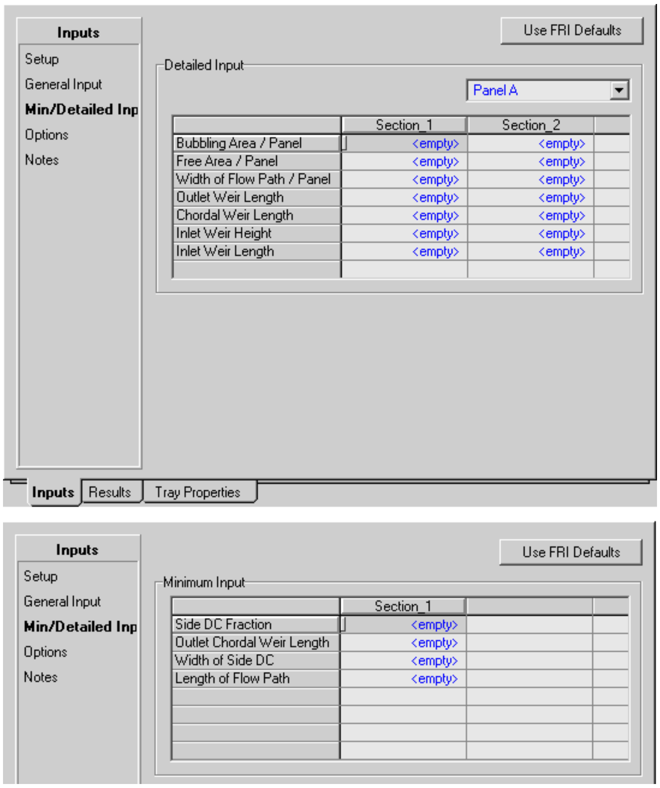
Object	Description
Tray Diameter row	Enables you to specify the tray diameter of the associate tray section.
Tray Spacing row	Enables you to specify the tray spacing of the associate tray section.
Use FRI Defaults button	Enables you to populate the general tray specifications with the FRI default values. The default values are obtained from the FRI spreadsheet.
Per Panel drop-down list	Enables you to select the panel you want to modify. The number of panels depend on the number of passes in the selected tray section: • For 1 pass, there is Panel A. • For 2 passes, there are Panel A and B. • For 3 passes, there are Panel A, B, and C. • For 4 passes, there are Panel A, B, C, and D.
Percent Hole Area row	Enables you to specify the percent value of the tray area that is made out of holes for the associate tray section.

Object	Description
Hole Diameter row	Enables you to specify the diameter of the holes in the associate tray section.
Tray Thickness row	Enables you to specify the tray thickness of the associate tray section.
Hole Edge Facing Vapour row	Enables you to select between **Sharp** or **Smooth** for the edges of the holes in the associate tray section.
Outlet Weir Height row	Enables you to specify the weir height for the associate tray section.
Side DC row	Enables you to specify the clearance height of the Side DC exit for the associate tray section.
Centre DC row	Enables you to specify the clearance height of the Side DC exit for the associate tray section. Only available if the tray section has two or more passes.
Off-Centre Near row	Enables you to specify the clearance height of the Side DC exit for the associate tray section. Only available if the tray section has three or more passes.
Off-Centre Far row	Enables you to specify the clearance height of the Side DC exit for the associate tray section. Only available if the tray section has three or more passes.

Min/Detailed Input Page

The Min/Detailed Input page enables you to specify the Minimum or Detailed Input variables of the selected tray section.

Figure 14.61



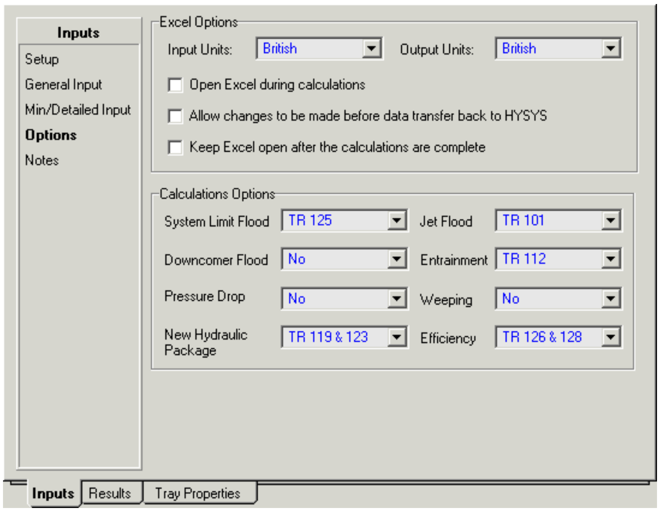
Object	Description
Use FRI Defaults button	Enables you to populate the Minimum or Detailed Input variables with the default values from the FRI spreadsheet.
Detailed Input group	This group is only available if you selected **Detailed Input** for the tray section configuration in the **Setup** page.
Panel drop-down list	Enables you to select the panel you want to modify.
Bubbling Area/Panel row	Enables you to specify the bubbling area of the associate section.

Object	Description
Free Area/Panel row	Enables you to specify the free area of the associate section.
Width of Flow Path/ Panel row	Enables you to specify the flow path width of the associate section.
Outlet Weir Length row	Enables you to specify the length of the outlet weir for the associate section.
Chordal Weir Length row	Enables you to specify the length of the chordal weir for the associate section.
Inlet Weir Height row	Enables you to specify the height of the inlet weir for the associate section.
Inlet Weir Length row	Enables you to specify the length of the inlet weir for the associate section.
Minimum Input group	This group is only available if you selected **Minimum Input** for the tray section configuration in the **Setup** page.
Side DC Fraction row	Enables you to specify the fraction of the side DC for the associate section.
Outlet Choral Weir Length row	Enables you to specify the length of the outlet chordal weir for the associate section.
Width of Side DC row	Enables you to specify the width of the side DC for the associate section.
Length of Flow Path row	Enables you to specify the length of the flow path for the associate section.

Options Page

The Options page enables you to calculation options of the FRI Tray Rating calculation.

Figure 14.62



Object	Description
Input Units drop-down list	Enables you to select the units for the input values in the FRI spread sheet.
Output Units drop-down list	Enables you to select the units for the output values in the FRI spread sheet.
Open Excel during calculations checkbox	Enables you to toggle between opening or hiding Excel during the FRI calculation.
Allow changes to be made before data transfer back to HYSYS checkbox	Enables you to toggle between allowing data changes before transferring the data into HYSYS or transferring the data straight to HYSYS as soon as the calculations are complete. For this option to work, you need to have the **Open Excel during calculations** checkbox selected.
Keep Excel open after the calculations are complete checkbox	Enables you to toggle between leaving Excel open or closing Excel as soon as the calculations are complete.
Calculation Options group	The calculation variables in this group is defined in the FRI spreadsheet.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page/Tab**.

The Notes page provides a text editor, where you can record any comments or information regarding the utility, or to your simulation case in general.

14.11.2 Results Tab

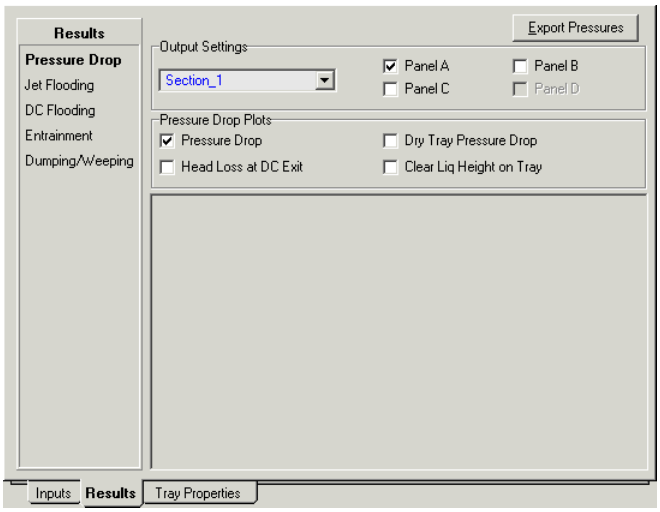
The Results tab displays the FRI Tray Rating calculations on the selected tray section, and these result values are grouped in the following pages:

- Pressure Drop
- Jet Flooding
- DC Flooding
- Entrainment
- Dumping/Weeping

Pressure Drop Page

The Pressure Drop page enables you to view the plot properties of a panel in a section.

Figure 14.63

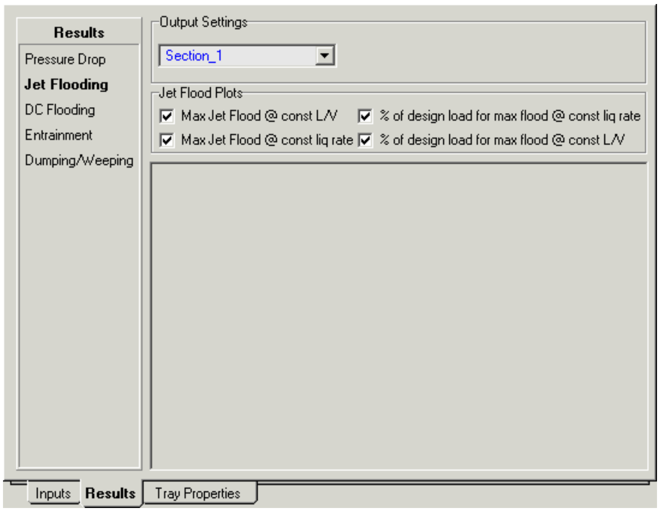


Object	Description
Export Pressure button	Enables you to export the plot data, calculated by FRI Tray Rating utility, back to the column in the PFD.
Output Settings drop-down list	Enables you to select the section you want to view in the plot.
Panel A checkbox	Enables you to toggle between displaying or hiding the plot data of Panel A.
Panel B checkbox	Enables you to toggle between displaying or hiding the plot data of Panel B.
Panel C checkbox	Enables you to toggle between displaying or hiding the plot data of Panel C.
Panel D checkbox	Enables you to toggle between displaying or hiding the plot data of Panel D.
Pressure Drop checkbox	Enables you to toggle between displaying or hiding the pressure drop data on the plot.
Head Loss at DC Exit checkbox	Enables you to toggle between displaying or hiding the head loss data on the plot.
Dry Tray Pressure Drop checkbox	Enables you to toggle between displaying or hiding the dry tray pressure drop data on the plot.
Clear Liq Height on Tray checkbox	Enables you to toggle between displaying or hiding the clear liquid height data on the plot.

Jet Flooding Page

The Jet Flooding page enables you to view the plot properties of the selected section.

Figure 14.64

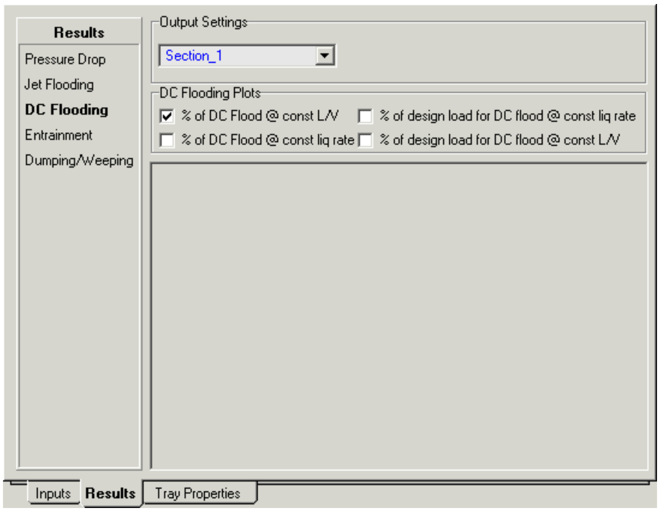


Object	Description
Output Settings drop-down list	Enables you to select the section you want to view in the plot.
Max Jet Flood @ const L/V checkbox	Enables you to toggle between displaying or hiding the maximum jet flood (at constant liquid vapour ratio) data on the plot.
Max Jet Flood @ const liq rate checkbox	Enables you to toggle between displaying or hiding the maximum jet flood (at constant liquid rate) data on the plot.
% of design load for max flood @ const liq rate checkbox	Enables you to toggle between displaying or hiding the design load % (at constant liquid rate) data on the plot.
% of design load for max flood @ const L/V checkbox	Enables you to toggle between displaying or hiding the design load % (at constant liquid vapour ratio) data on the plot.

DC Flooding Page

The DC Flooding page enables you to view the plot properties of the selected section.

Figure 14.65

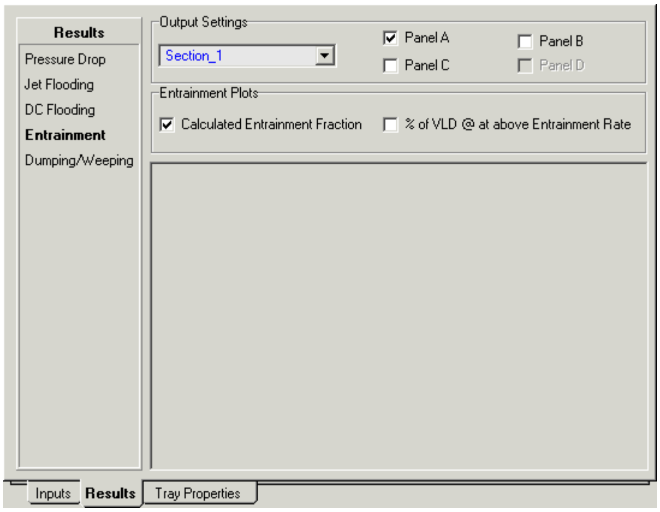


Object	Description
Output Settings drop-down list	Enables you to select the section you want to view in the plot.
% of DC Flood @ const L/V checkbox	Enables you to toggle between displaying or hiding the DC flood % (at constant liquid vapour ratio) data on the plot.
% of DC Flood @ const liq rate checkbox	Enables you to toggle between displaying or hiding the DC flood % (at constant liquid rate) data on the plot.
% of design load for DC flood @ const liq rate checkbox	Enables you to toggle between displaying or hiding the DC design load % (at constant liquid rate) data on the plot.
% of design load for DC flood @ const L/V checkbox	Enables you to toggle between displaying or hiding the DC design load % (at constant liquid vapour ratio) data on the plot.

Entrainment Page

The Entrainment page enables you to view the plot properties of a panel in a section.

Figure 14.66

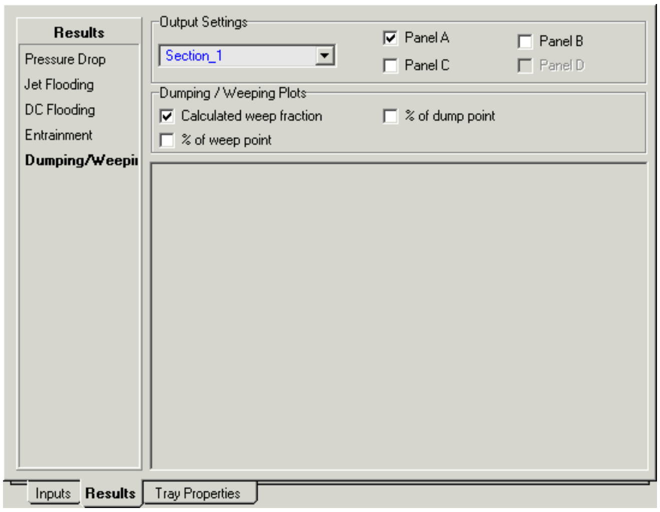


Object	Description
Output Settings drop-down list	Enables you to select the section you want to view in the plot.
Panel A checkbox	Enables you to toggle between displaying or hiding the plot data of Panel A.
Panel B checkbox	Enables you to toggle between displaying or hiding the plot data of Panel B.
Panel C checkbox	Enables you to toggle between displaying or hiding the plot data of Panel C.
Panel D checkbox	Enables you to toggle between displaying or hiding the plot data of Panel D.
Calculated Entrainment Fraction checkbox	Enables you to toggle between displaying or hiding the calculated entrainment fraction data on the plot.
% of VLD @ above Entrainment Rate checkbox	Enables you to toggle between displaying or hiding the VLD % (above the entrainment rate) data on the plot.

Dumping/Weeping Page

The Dumping/Weeping page enables you to view the plot properties of a panel in a section.

Figure 14.67



Object	Description
Output Settings drop-down list	Enables you to select the section you want to view in the plot.
Panel A checkbox	Enables you to toggle between displaying or hiding the plot data of Panel A.
Panel B checkbox	Enables you to toggle between displaying or hiding the plot data of Panel B.
Panel C checkbox	Enables you to toggle between displaying or hiding the plot data of Panel C.
Panel D checkbox	Enables you to toggle between displaying or hiding the plot data of Panel D.
Calculated weep fraction checkbox	Enables you to toggle between displaying or hiding the calculated weep fraction data on the plot.
% of weep point checkbox	Enables you to toggle between displaying or hiding the weep point % data on the plot.
% of dump point checkbox	Enables you to toggle between displaying or hiding the dump point % data on the plot.

14.11.3 Tray Properties Tab

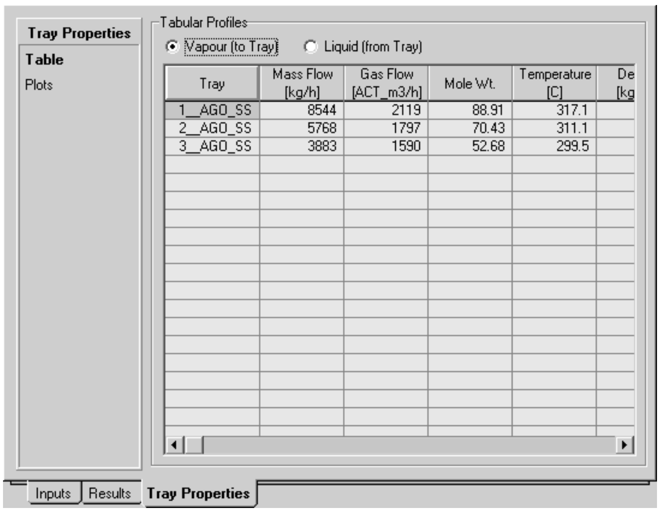
The Tray Properties tab contains individual tray property information of the trays in the selected tray section and these information are grouped in the following pages:

- Table
- Plots

Table Page

The Table page displays the vapour or liquid tray properties in a tabular format.

Figure 14.68



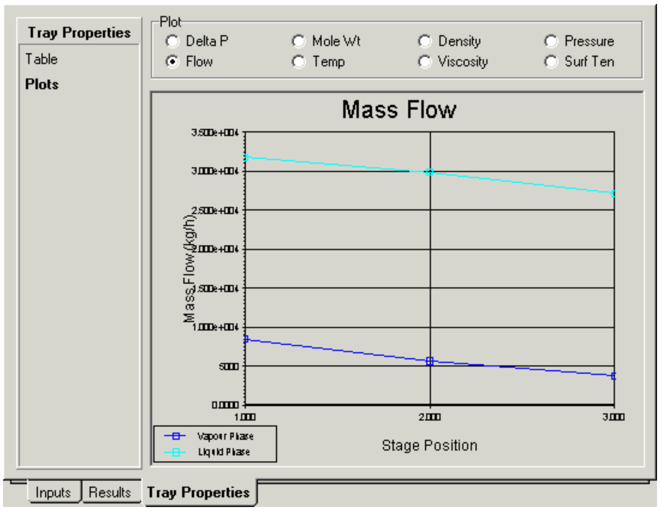
Object	Description
Vapour (to Tray) radio button	Enables you to view the following properties on the vapour entering the tray: • Mass Flow • Temperature • Gas Flow • Density • Molecular Weight • Viscosity

Object	Description
Liquid (from Tray) radio button	Enables you to view the following properties on the liquid leaving the tray: • Mass Flow • Liquid Flow • Molecular Weight • Temperature • Density • Viscosity • Surface Tension

Plots Page

The Plots page displays the selected tray property data in plot format.

Figure 14.69



Object	Description
Delta P radio button	Displays the pressure drop between the trays in the selected tray section.
Flow radio button	Displays the liquid and vapour flow rates within the trays in the selected tray section.
Mole Wt radio button	Displays the liquid and vapour molecular weights within the trays in the selected tray section.
Temp radio button	Displays the liquid and vapour temperatures within the trays in the selected tray section.
Density radio button	Displays the liquid and vapour densities within the trays in the selected tray section.

Object	Description
Viscosity radio button	Displays the liquid and vapour viscosities within the trays in the selected tray section.
Pressure radio button	Displays the vapour pressure within the trays in the selected tray section.
Surf Ten radio button	Displays the liquid surface tension within the trays in the selected tray section.

14.12 Hydrate Formation Utility

To add the Hydrate Formation utility, refer to the section on **Adding a Utility**.

The Hydrate Formation utility calculates the incipient solid formation point for hydrates. The predictive models are based on fundamental thermodynamic principles and use equation-of-state generated properties in calculating the equilibrium conditions. These predictive models can therefore be applied to various compositions, and extreme operating conditions with a greater degree of reliability than one may expect with empirical expressions or charts.

Hydrate formation prediction is restricted to the Peng-Robinson and Soave-Redlich-Kwong equations of state.

A hydrate curve can be plotted with the Envelope utility.

Hydrate Calculation Models

The only requirement for hydrate formation is that some water must be present in either the vapour or condensed hydrocarbon phase with hydrate forming components. Once favourable pressure and temperature conditions are reached (high pressures or low temperatures), the mixture of hydrate-forming molecules and water molecules form a non-stoichiometric solid phase.

These favourable conditions can be well above the freezing point of water, or well before the point where free water or ice would drop out.

The hydrate formers are limited to molecules that are small enough to fit into the cavities formed by the host water lattice structure. These include low molecular weight paraffinic hydrocarbons up to n-butane, some olefins, and some of the smaller non-hydrocarbon components such as carbon dioxide, nitrogen, oxygen, argon, and hydrogen sulphide.

The model used for predicting the incipient hydrate point for hydrates in equilibrium with free water is based on the original equilibrium model proposed by van der Waals and Platteeuw[11] and later modified by Parrish and Prausnitz[8]. The same model has been incorporated and enhanced by AspenTech for its hydrate predictions. The equation of state is used to predict the properties of the hydrate-forming components in equilibrium with the solid hydrate phase.

The Calculation Mode field on the Design and Performance tabs in the hydrate utility has been restored as in previous HYSYS versions. The calculation modes reported are Vapour phase, Liquid phase, Free water found, and Assume Free Water.

The four hydrate calculation modes/scenarios and the appropriate model treatment are described as follows:

- **Vapor Phase Hydrates**

  For scenarios that result in no free aqueous phase after an equilibrium flash, in other words Vapor only, Vapor-Liquid, and Vapor-Liquid-Liquid (Liquid refers to a hydrocarbon liquid) cases, the Vapor phase hydrates are obtained and the Vapor model is used for hydrate prediction as long as there is appreciable amount of the vapor phase.

  The Vapor Model is based on the work of Ng and Robinson[6]. The fugacity of water, as a function of pressure and temperature in the empty lattice (MT), is determined by data reduction. Plots of $\ln f_{w,o}$ vs. $1/T$ and of $(d\ln f_w)/(dP)$ vs. T show linear relationships.

where:

$f_{w,o}$ = *fugacity of the water at zero pressure over the un filled structure II lattice*

The empty lattice water fugacity at any pressure is represented by the following expression:

$$\ln f_w^{MT} = \ln f_{w,\,o}^{MT} + \left(\frac{d\ln f_w^{MT}}{dP}\right)_T P \tag{14.20}$$

where:

f_w^{MT} *= empty lattice fugacity at any pressure*

$f_{w,o}$ *= fugacity of the water at zero pressure*

P = pressure.

By combining this expression with the linear regressed plots, the fugacity of water over the unfilled hydrate lattice as a function of temperature and pressure is obtained. The relationships depend on hydrate structure but are independent of the composition of the examined mixture.

For hydrates of Structure I, the fugacity relationships are found to be:

$$\ln f_{w,\,o}^{MT} = 14.269 - \frac{5393}{T} \tag{14.21}$$

$$\left(\frac{d\ln f_w^{MT}}{dP}\right)_T = 0.00036T - 0.1025 \tag{14.22}$$

where:

T = temperature in Kelvin

For hydrates of Structure II, the fugacity relationships are found to be:

$$\ln f_{w,\,o}^{MT} = 18.062 - \frac{6512}{T} \tag{14.23}$$

$$\left(\frac{d\ln f_w^{MT}}{dP}\right)_T = 0.0001109T - 0.03192 \tag{14.24}$$

where:

T = temperature in Kelvin

- **Free Water Found**

  For scenarios that result in a free aqueous phase after an equilibrium flash, in other words Aqueous only, Vapor-Aqueous, Liquid-Aqueous, and Vapor-Liquid-Aqueous (Liquid refers to a hydrocarbon liquid) cases, free water hydrates are obtained and the free water model (or Symmetric Model) is used for hydrate prediction as long as there is appreciable amount of the aqueous phase.

  The Free Water Model is based on the work of Ng and Robinson[3]. The Parrish-Prausnitz algorithm is modified to allow for the prediction of hydrates in aqueous systems. All fluid properties including phase behavior, volumetric behavior, and fugacities are calculated with the selected equation of state. The Kihara parameters for each hydrate-forming component are recalculated based on the work by Ng and Robinson.
- **Assume Free Water**

  In the absence of water as a component in the simulation or when the amount of water in the stream being analysed equals zero, the stream is saturated with water and the free water model described above is used for hydrate predictions.
- **Liquid Phase Hydrates**

  For scenarios that result in no free aqueous phase or vapor phase after an equilibrium flash, in other words Liquid only and Liquid-Liquid (Liquid refers to a hydrocarbon liquid) cases, Liquid phase hydrates are obtained and the Vapor model described above is used for hydrate prediction. Results from this model at best describe hydrate predictions for this undersaturated case.

The Hydrate Formation utility in HYSYS has been improved by restoring the default calculation model for prediction of hydrates. In other words, HYSYS automatically determines the appropriate model to be used based on the result of the equilibrium flash. The default calculation model is recommended for all the hydrate prediction scenarios described above.

If you want to have control over model selection (not generally recommended), access the Model Override page, select the Override Default Model checkbox, and select the appropriate model from the Hydrate Calculation Model drop-down list for performing hydrate prediction calculations.

The four calculation models in the Model Override page are:

- Assume Free Water
- Asymmetric
- Symmetric
- Vapour Only

Each of the calculation models is described in the following sections and will be used only if you choose to override the default model on the Model Override page.

Assume Free Water

The Assume Free Water model calculates the hydrate formation temperature by assuming the stream is at the saturation point of water at hydrate conditions, neglecting the amount of water present in the stream.

If the hydrate results are calculated for a stream with no water in it (in other words, the default model is Assume Free Water) and the same stream with water, manually selecting the calculation model to be Assume Free Water, the hydrate results will be very similar. The difference is due only to the composition difference between the streams when water is removed.

Asymmetric

The Asymmetric model automatically switches between the two subset-models (Symmetric and Vapour Only) according to the presence of the phases determined by the flash calculation at hydrate conditions.

Symmetric

The Symmetric model uses parameters fitted from experimental data representing a wide variety of hydrate systems. The experimental data used to determine these parameters are for liquid/aqueous-hydrate systems. This model uses the free water model formulation as described in the **Free Water Found** section, and is therefore appropriate for systems with a free-water phase.

Vapour Only

The Vapour Only model uses parameters fitted from experimental data for vapour-hydrate systems. This model uses the Vapor model as described in the **Vapor Phase Hydrates** section, and is therefore appropriate for systems without a free-water phase (vapor hydrate and liquid [hydrocarbon] hydrate).

Both the Symmetric and Vapour Only models are subsets of the Asymmetric model. These models provide the user more access to the choices of the calculation models, and can be useful when the Asymmetric model fail to find the correct phases at hydrate conditions. For example, the Asymmetric model may not be able to choose the correct phase near a phase boundary, and thus these two models can be used to force the hydrate utility to choose the correct phase at the hydrate conditions.

With the exception of the Assume Free Water model, when a calculation model is chosen, the final Calculation Mode field (found on the Performance tab of the Hydrate utility property view) must be checked to ensure that the equilibrium phase and the hydrate model chosen are consistent. For example, if the Symmetric model is chosen for calculations, and the equilibrium phase determined by the flash is found to be of vapour type, the Asymmetric or Vapour Only model should be chosen for this system.

When cases containing Hydrate utilities are loaded from previous versions, the revised calculation model is automatically selected (as in previous HYSYS versions) and is used for hydrate predictions.

If you want to have flexibility over the model selection namely Assume Free Water, Asymmetric, Symmetric, and Vapour only, you can override the model by accessing the Model Override page and then selecting the appropriate above-mentioned model.

Ice Formation

In the case of ice formation, HYSYS displays “Ice Forms First” in the Hydrate Type Formed field, and the hydrate formation temperature and/or pressure are set to <empty>.

14.12.1 Design Tab

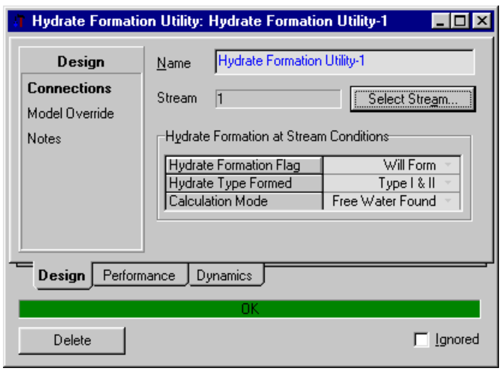
The Design tab contains the following pages:

- Connections
- Model Override
- Notes

Connections Page

On the Connections page, you can connect a stream to the Hydrate Formation utility, and change the utility’s name.

Figure 14.70



Click the Select Stream button to open the Select Process Stream property view. On the Select Process Stream property view, you can select the stream you want to be connected to the utility.

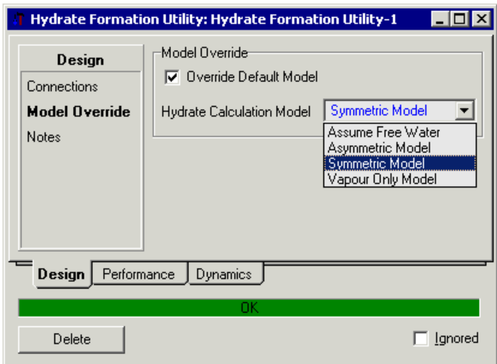
The Hydrate Formation status at the current stream conditions are also shown on the Connections page.

Hydrate Formation Status	Description
Hydrate Formation Flag	Displays the status of hydrate formation. There are two possibilities: • **Will Form** • **Will NOT Form**
Hydrate Type Formed	Displays the types of Hydrate formed. It is possible that Ice forms first, in which case HYSYS displays the message **Ice Forms First** in the appropriate field. If the temperature is higher than the formation temperature, then **No Types** is displayed in this field.
Calculation Mode	Possibilities are Vapour Phase, Liquid Phase, Free Water Found, or Assume Free Water. HYSYS can predict the incipient solid formation point for systems consisting of gas hydrates in equilibrium with a free-water phase, or for systems without a free-water phase. When you choose to override the default model, you should check that the final Calculation Mode is consistent with the model chosen. It is not necessary for a free-water phase to be present for hydrate formation. For either case, the correct model is used to predict the incipient point for solid hydrates. If water is not specified as a component, HYSYS assumes the stream to be saturated with water.

Model Override Page

The Model Override page allows you to gain control over a specific model for hydrate predictions. You can override the default model by selecting the **Override Default Model** checkbox and selecting the appropriate model for hydrate calculations as shown in the figure below.

Figure 14.71



Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

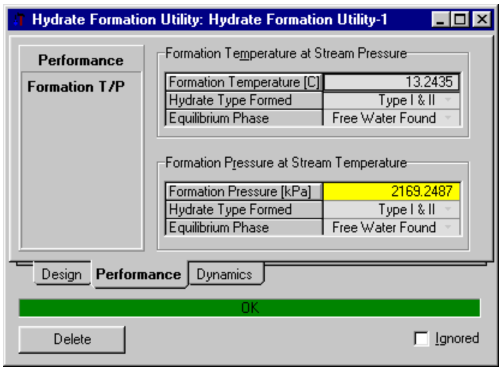
The Notes page provides a text editor, where you can record any comments or information regarding the utility, or to your simulation case in general.

14.12.2 Performance Tab

The Performance tab contains one page, Formation T/P. The Formation T/P page contains two groups:

- Formation Temperature at Stream Pressure
- Formation Pressure at Stream Temperature

Figure 14.72



Formation Temperature at Stream Pressure Group

The hydrate types and calculation models are discussed in the **Hydrate Calculation Models** section.

The Formation Temperature at Stream Pressure group displays the formation temperature at which hydrates are formed at the stream pressure. The hydrate type and equilibrium phase are also shown for the hydrate, which would form at this formation temperature.

Formation Pressure at Stream Temperature Group

The hydrate types and calculation models are discussed in the **Hydrate Calculation Models** section.

The Formation Pressure at Stream Temperature group displays the formation pressure at which hydrates are formed at the stream temperature. The hydrate type and equilibrium phase are also shown for the hydrate, which would form at this formation pressure.

Hydrate Inhibition

To avoid or inhibit the formation of hydrates, you can either set the operating conditions to be outside the predicted equilibrium curve for hydrates, or inject inhibitor solvents such as glycols or alcohols to suppress the formation of hydrates. The solvents serve as antifreeze agents, and depress the freezing conditions of hydrates.

To inhibit the formation of hydrates of a given stream in the flowsheet, you must install a stream containing the solvent, for example, either methanol or glycol. Use the Mixer operation to mix it with the process stream, and then access the Hydrate utility to find the new solid hydrate formation condition. HYSYS also reports if the solid solvent phase forms before the hydrates form, in other words, solid methanol or glycol solution. These equilibrium conditions are all fitted from known phase diagrams. The eutectic point formed by the solvent mixture results in a solid methanol or glycol phase if a high concentration is used.

In setting up a Flowsheet for hydrate inhibition, ensure the conditions of the solvent injection stream are all sufficiently defined (in other words, the temperature, pressure, flow rate, and composition are specified) so the property package can flash the mixed stream. As a result of solvent injection, the hydrate-forming conditions are reduced due to association of the inhibitor with the water in the current phase, be it vapour or liquid phase.

Since three phase thermodynamics are used to perform the flash calculation, the phase distribution of the components, including water and the solvent, can be calculated rigorously. Therefore, solvent losses in the hydrocarbon liquid and vapour phases are properly taken into account.

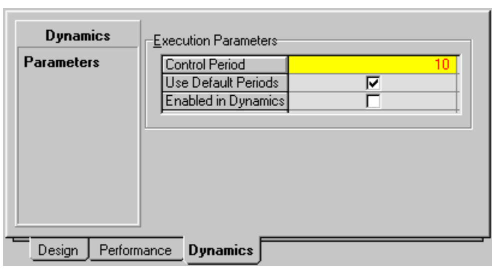
The PR equation of state was not originally designed for non-ideal components such as methanol and glycols. Ensure the resulting distribution of the components in all phases is satisfactory, especially if three phases exist. The solubility of methanol in the hydrocarbon and aqueous phases is optimized with the PR Equation of State for the methanol-HC-water VLE. Make further adjustment to the PR interaction parameters to meet your own specifications.

Overall, this approach should be more accurate than using Hammerschmidt's equation which was developed more for dilute solutions of antifreeze agents. The Hammerschmidt equation applies only for typical natural gas mixtures and for solute concentrations less than 20 mole per cent. Although it is applied for cases beyond this region with reasonable success, this is attributed to a number of compensating factors. For validation of this model, refer to **GPA Research Report RR-66**.

14.12.3 Dynamics Tab

The Dynamics tab allows you control how often the utility gets calculated when running in Dynamic mode.

Figure 14.73



The Control Period field is used to specify the frequency that the utility is calculated. A value of 10 indicates that the utility be recalculated every 10th pressure flow step. This helps speed up your dynamic simulation since utilities can require some time to calculate.

The **Use Default Periods** checkbox lets you set the control period of one utility to equal the control period of any other utilities that you have in the simulation. For example, if you have five utilities and require them all to have a control period of 5 and currently the value is 8, with this checkbox selected if you change the value in one utility all the other utilities change. Alternatively, if you want all the utilities to have different values, then clear this checkbox.

The **Enable in Dynamics** checkbox is used to activate this feature for use in Dynamic mode.

14.13 Master Phase Envelope Utility

The Master Phase Envelope Utility allows you to calculate the three-phase envelope for multiples streams of known compositions, including streams with only one component.

14.13.1 Design Tab

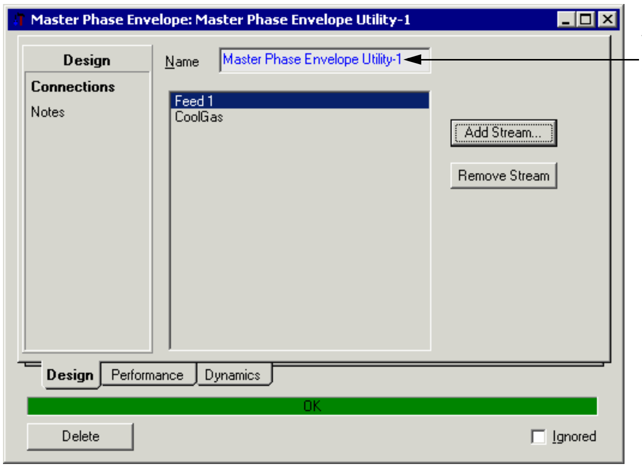
The Design tab contains the following pages:

- Connections
- Notes

Connections Page

On the Connections page, you can add or remove streams to be used in the phase envelope calculations.

Figure 14.74



You can change the utility's name by typing in the Name field.

Click the Add Stream button to open the Select Process Stream property view. On the Select Process Stream property view, you can select the stream you want to be connected to the utility. You can remove a stream from the list by selecting the stream and clicking the Remove Stream button.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor, where you can record any comments or information regarding the utility, or to your simulation case in general.

14.13.2 Performance Tab

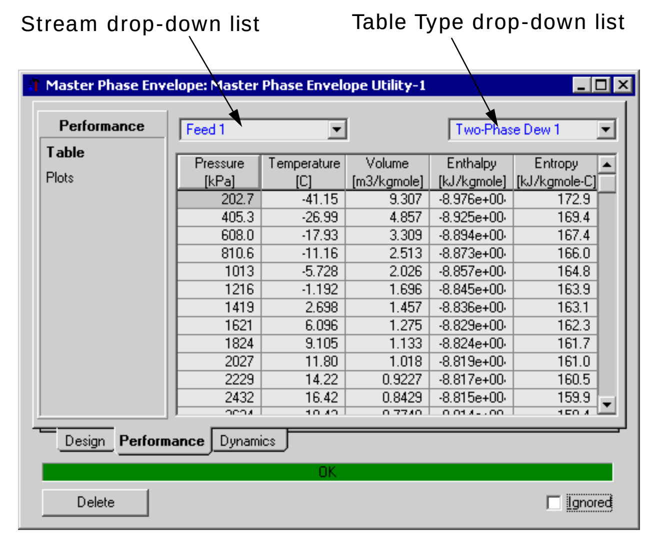
The Performance tab contains the following pages:

- Table
- Plots

Table Page

You can view the envelope results in tabular format on the Table page.

Figure 14.75



The Table page allows you to display the envelope results for one stream at a time.

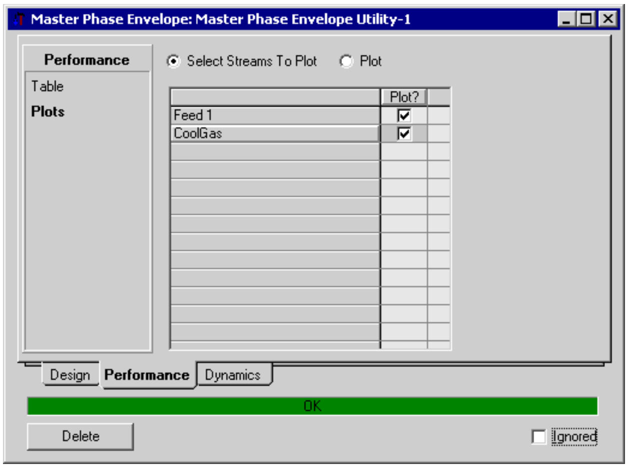
To view the tabular results of different envelopes for the desired stream, select a stream from the Stream drop-down list then select a table type from the Table Type drop-down list. All Isocurves and Quality lines associated with the individual envelopes are transferred to the table.

Plots Page

The Plots page allows you to display different types of envelope graphs for multiple streams.

You can plot the envelope graphs for the desired stream(s) by clicking the Select Streams To Plot radio button. All the streams that have been attached to the utility are listed in the table.

Figure 14.76

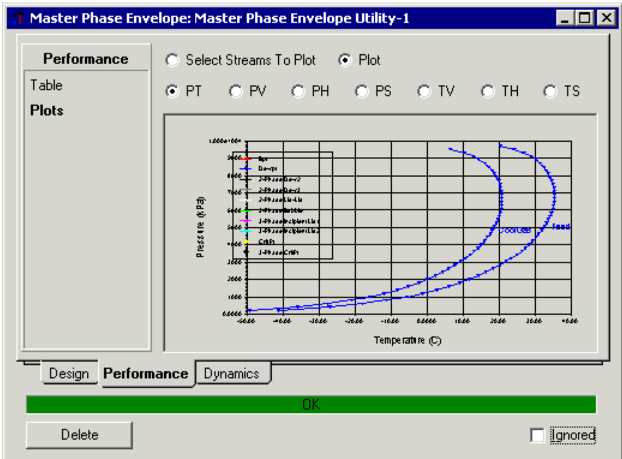


Click the Plot radio button to plot the selected streams. Depending on the radio button selected, the vapour-liquid, liquid-liquid, and vapour-liquid-liquid envelopes can be plotted for the following variables:

- Pressure-Temperature
- Pressure-Volume

- Pressure-Enthalpy
- Pressure-Entropy
- Temperature-Volume
- Temperature-Enthalpy
- Temperature-Entropy

Figure 14.77



14.14 Parametric Utility

The Parametric Utility enables you to create neural networks to replace portions of the simulation flowsheet. You can now easily configure the utility to capture data from the flowsheet model. You can define a list of variables that you want to perturb (manipulated variables), and variables you want to record (observable variables). The utility allows you to quickly create lists of variables and to re-use variable lists. The data generated can be exported in a comma-delimited style in a number of different formats.

The utility originated as a set of tools for building a Parametric model (PM) within the HYSYS environment. The utility integrates Neural Network (NN) technology into its framework. The major function of the Parametric utility is to approximate an existing HYSYS model with a Parametric model.

Using a Parametric model with neural network capability to approximate a HYSYS model significantly improves the robustness of the model, and reduces its calculation time thereby improving overall performance. The accuracy of the model depends upon the data available, and the type of model being approximated.

The object of analysis can be a collection of unit operations, an entire flowsheet, or a number of selected variables. Using input and output data sets as training data, the neural network algorithm determines the Parametric model parameters. This step is called training but can also be referred to as regression or identification.

Steps one to four describe the general procedure for the Utility:

1. Select scope.
2. Select variables (manipulated and observable).
3. Define test datasets.
4. Generate data.

Steps five to six describe the general procedure to use for NN:

5. Train.

6. Validate.

The Parametric Utility main purpose is to generate data and training for Neural Networks. This includes setting up the utility, and generating the data with some additional steps used in building the NN.

14.14.1 Neural Networks

What is a Neural Network?

A Neural Network (strictly 'Artificial Neural Network' as opposed to a 'Biological Neural Network') is a mathematical system with a structure based on 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

Neural Networks provide a performance and cost effective modeling tool, and can extend the capabilities of traditional statistics, modeling, and control. They can be applied in both linear and non-linear systems where first-principles modeling is too costly or difficult.

Neural Networks provide flexible and powerful techniques for data analysis, and can be used for:

- dynamic and static process modeling
- nonlinear and adaptive control
- inferential predictions
- time series prediction
- multivariate pattern recognition

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 modelled using the first principles.

Using a Neural Network solver offers a number of advantages:

- It is significantly faster that 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.

There are three parts to the HYSYS Neural Network implementation:

- **Parametric Utility**. Where the Neural Network is configured and trained.
- **Parametric Unit Operation**. Allows the Neural Network to appear as a unit operation on the flowsheet, and it is typically used when taking a "black box" approach.
- **Neural Network Manager**. Allows you to switch Neural Network (NN) Objects into appropriately configured Parametric Utilities, and to generate simple Neural Network's from external data.

Refer to **Section 14.14.4 - Neural Network (NN) Manager** for more information on the Neural Network Manager.

14.14.2 Variables

Refer to **Section 5.6 - Parametric Unit Operation** for details on the Parametric unit operation.

The parameters of the Parametric model are determined either through HYSYS simulation runs or based on historical plant data (the latter also requires the use of the Parametric unit operation). There are three types of variables:

- Observable
- Manipulated
- Training

The variables are discussed in the following sections.

Observable Variables

Observable variables can be either input or output variables within the HYSYS PFD model. When HYSYS is used to generate training datasets for the Parametric model, a number of simulation runs must be performed. During the simulation run, the simulation solution engine calculates each operation in the HYSYS PFD. The observable variables are the HYSYS variables whose values are known, and used as training data when calculating the Parametric model.

Observable input and output variables can each include both input and output stream variables. A HYSYS model parameter with a varying value can be either an observable input variable or an observable output variable within the Parametric model.

Manipulated Variables

Manipulated variables are the variables being modified in the Parametric utility, and are obtained from the HYSYS PFD model simulation.

Training Variables

Training variables are a combination of both the observable and manipulated variables used to develop the Parametric model. The term “training” refers to the task of using the data sets available as a form of “learning” that, in effect, fits the model parameters to the specifications.

The Parametric model approximates the HYSYS model in the sense that, given the same values for the training input variables, the values of the output variables from the Parametric model must be close to the values of the output variables from the HYSYS model.

There are no methods for training neural networks that can “create” information that is not contained in the training data. The neural network model is only as good as its training data.

14.14.3 PM Utility Property View

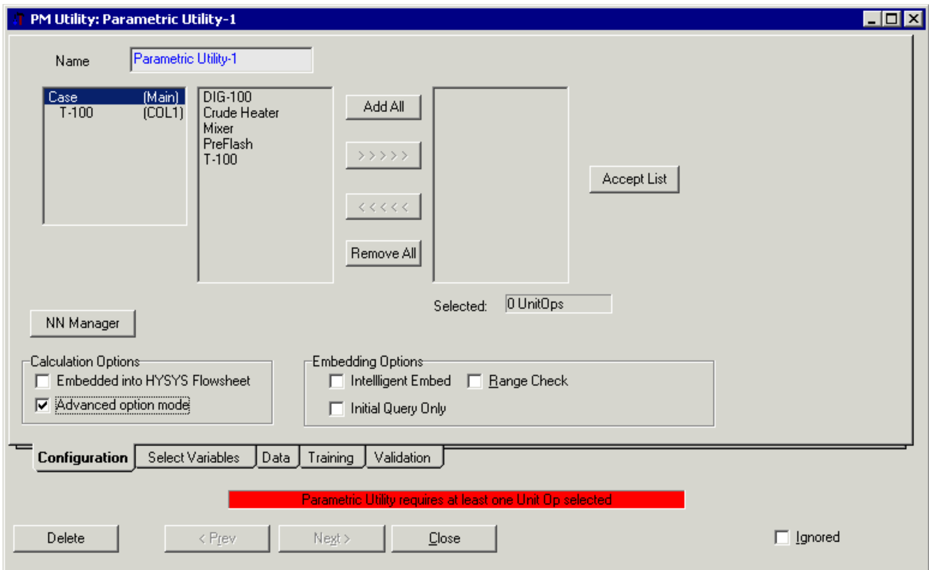
To add the Parametric Utility, refer to the section on **Adding a Utility**.

The PM Utility property view is composed of several tabs that are described in the following sections. The status bar at the bottom of the screen provides you with hints and descriptions of what the fields represent.

Configuration Tab

The Configuration allows you to specify the name of the utility, calculation options, and select HYSYS objects to be included in the Parametric model.

Figure 14.78



The Configuration tab defines the scope of the utility and lists the equipment from which you can select objects to configure. Once objects have been added to the final list, you must click the **Accept List** button and then click the **Next** button to proceed**.**

There are two Calculation Options available which are described below:

Calculation Options	Description
Embedded into HYSYS Flowsheet	This checkbox allows the trained neural network to replace the traditional HYSYS solver for the objects included in the NN.
Advanced option mode	This checkbox enables more flexibility in the selection of manipulated variable sets. If selected, the Embedding option checkboxes are displayed. • Intelligent Embed - For embedding, it is required that the stream variables selected are a flashable set. If checked, and the streams are over-specified, the utility does not query the user and selects a sub-set of variables that allows the streams to flash. If the streams are under-specified, the utility will not warn the user. • Initial Query Only - The above queries occur only once on the initial embedding unless the PMUtil configuration is changed. • **Range Check -** If the manipulated variable values are outside the range than the Neural Network in the Parametric Utility was trained upon, then the Parametric Utility is unembedded.

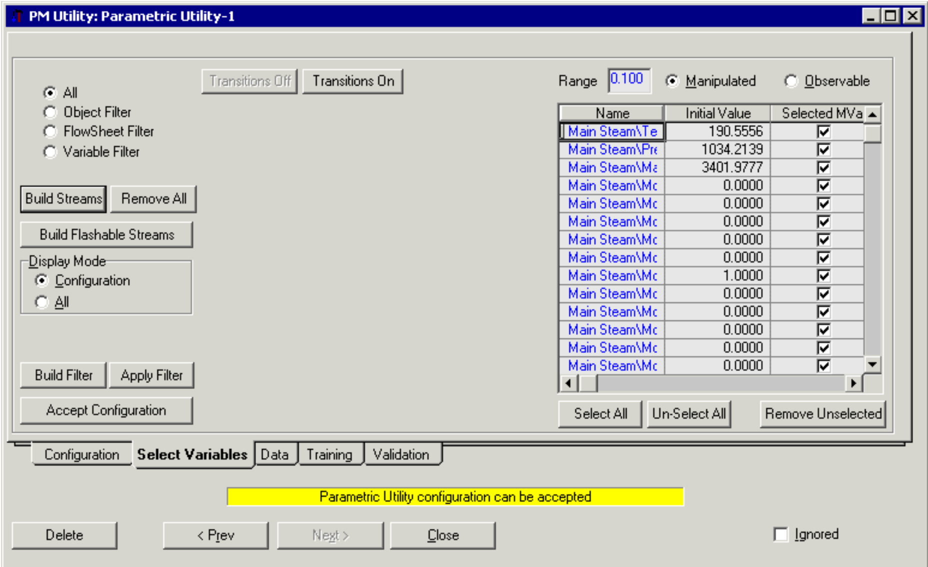
You cannot advance to the next tab (Select Variables) unless you click the Accept List button.

The Accept List button accepts the changes and obtains all variables known to the selected objects from the HYSYS flowsheet.

Select Variables Tab

The Select Variables tab allows you to filter your objects so that you can add manipulated and observable variables. The various functions on this tab are described below.

Figure 14.79



When you select the Manipulated radio button, a group of radio buttons become available for you to select a filter type from. This is to help you choose your manipulated and observable variables:

- **All**. Shows all chosen variables of the selected type in the table. If you didn't choose any, none will appear.
- **Object Filter**. Shows a list of unit operations and objects in a tree browser for selection.
- **FlowSheet Filter**. Shows a list of subflowsheets contained in the case and any related variables.
- **Variable Filter**. Shows all flowsheet variables.

The Property Filter radio button only appears if the Observable radio button is selected.

The table below describes the available objects on the Select Variables tab.

The following buttons/radio buttons/fields are not always visible. Certain ones appear when the Manipulated or Observable radio buttons are selected, and others appear when you click the Change/Accept Configuration button.

Object	Description
Build Streams button	Builds variables based on the stream information in the utility scope. It selects all variables that can be modified in the streams as Manipulated variables and all stream variables as Observable Variables (often duplicating). If you have already created a set of variables, you must click the Change Configuration button to see this button.
Remove All button	Removes all variables from the table. If you have already created a set of variables, you must click the Change Configuration button to see this button.
Build Flashable Streams button	Selects all variables in the streams that can be modified and sets them as manipulated. For observable variables, it takes as many variables as necessary from the streams to flash without a inconsistency error. The flash used is a T/P/F flash.
Display Mode group	The group contains two radio buttons to choose from: • Configuration • All The radio buttons control how many columns appear in the variables table, and what type of data is displayed.
Build Filter button	Allows you to create a filter that adds all objects in the utilities scope that match your filter criteria. For example, if you want every stream temperature, pressure, and volume flow, you can build a filter to add all streams with these three variables. You can save and reuse these filters. If you have already created a set of variables, click the Change Configuration button to see this button.
Apply Filter button	Once a filter is built, click this button to add all variables that meet the filter criteria. Click the Change Configuration button to see this button.
Change/Accept Configuration buttons	You must click the Change Configuration button when you want to change pre-selected variables. The Change Configuration button then becomes the Accept Configuration button. Any new changes will be updated when you click the Accept Configuration button.

Object	Description
Range field	If you change the Range, then the Low Limit/High Limit of the Manipulated Variables changes. For example, a value of 0.1 gives you a Low-High of $\pm 10\%$. The Low Limit and High Limit are max/min used when generating random values for the manipulated variables on the Data tab. This is important when you want to randomly select the manipulated variables commonly applied with neural networks. This field is only visible when the Manipulated radio button is selected.
Manipulated radio button	Displays manipulated variable property view.
Observable radio button	Displays observable variable property view.
Select All button	With either the observable or manipulated variables in the table, this button selects all checkboxes in the active list. Click the Change Configuration button to see this button.
Un-Select All button	With either the observable or manipulated variables in the table, click this to unselect all checkboxes in the active list. Click the Change Configuration button to see this button.
Remove Unselected button	Removes all unselected variables from either the observable or manipulated tables. Click the Change Configuration button to see this button.
Transitions Off / On buttons	The Transition On/Off buttons only appear when stream cutters are used, and you have selected the Advanced option mode. Stream cutters are inserted into the boundary streams when a subset of the flowsheets objects is selected.

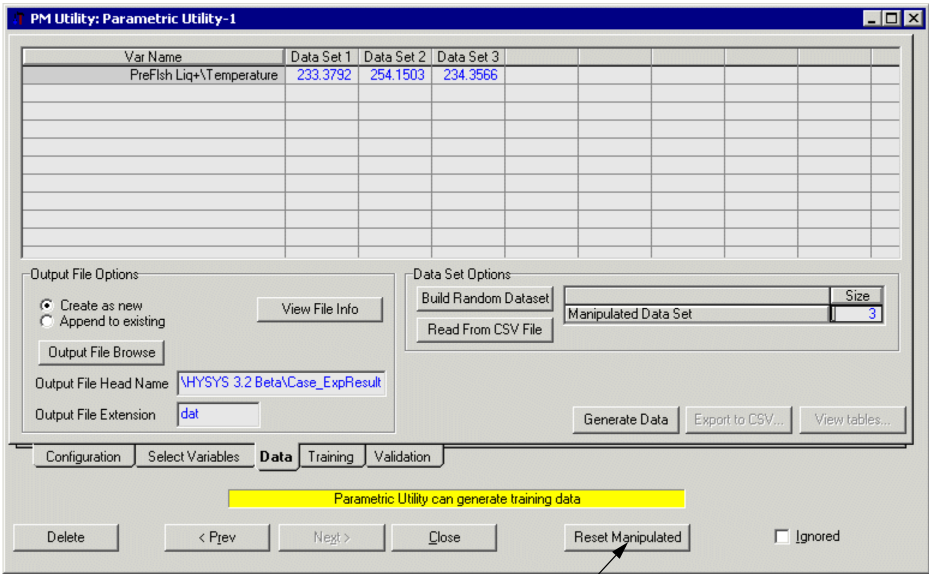
Refer to **Section 5.11 - Stream Cutter** for more information.

When you are satisfied with your selection of observable and manipulated variables, click the **Accept Configuration** button. The current variable configuration is accepted, and you can now access the Data tab.

Data Tab

The Data tab allows you to configure and generate input and output data sets for the Parametric model based on HYSYS simulations. Training data sets are generated by using stepwise changes to the manipulated input variables to produce varying output results.

Figure 14.80



Used to reset manipulated variables to their original values after data generation.

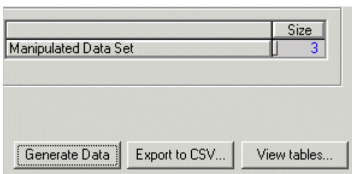
The table below describes the objects on the Output File Options group and Data Set Options group.

Object	Description
Output File Options group	
Create as new radio button	Create as new allows you to name and create a new file to store your data. This is a binary file that is used internally by the PM Utility and cannot be read in as a CSV file. When sharing a case in the PM utility, and the data has been generated, make sure the binary data file is also transferred, otherwise the data will have to be re-generated.

Object	Description
Append to existing radio button	Append to existing saves your data to an existing *.dat file. This is useful when you want to append new data to previously generated data.
Output File Browse button	Click to browse for a *.dat file.
View File Info button	Displays a property view showing the *.dat file info, such as maximum file size, total number of records, and so forth.
Output File Head Name field	The path for the output file is specified here.
Output File Extension field	Specify the type of file you want to save the output file as.
Data Set Options group	
Build Random Dataset button	Picks random manipulated variables between high and low bounds for each variable.
Read from CSV File button	Click to browse for a *.csv file that you can import. It contains manipulated set points from a comma delimited file instead of entering data manually. In this file a line that begins with a * is taken as a comment line.

The Manipulated Variable Set matrix is shown below with its corresponding buttons.

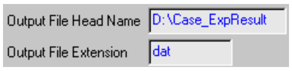
Figure 14.81



The table below describes the objects on the Manipulated Variable set matrix.

Object	Description
Size column	Enter the number of manipulated variable data sets you want to run. When you change this number, the number of data set columns associated with the variable list changes.
Generate Data button	Initiates the simulation engine to generate training data for the parametric model based on the HYSYS model.

Object	Description
Export to CSV... button	Once data has been generated, you can save manipulated and observable data to a *.csv file; you can choose between a number of formats.
View tables... button	Views the manipulated and observable variables in the HYSYS environment. This is only available when the Advanced Option Mode is selected.



When setting up your Parametric model for the first time, click the **Create as new** radio button. This allows you to name and create a new file to store your data. Data is then written to an external file based on the default name and location (path) listed.

Later, if you want to add to the number of data sets used for training (thereby increasing the accuracy of your Parametric model), choose the **Append to existing** radio button. Data is then written to an external file based on the default name and location listed (as shown in the previous figure).

If you have changed the model's configuration, you should not append to existing data sets.

The Output File Options group displays information related to the training data file to be generated by the Parametric Utility. The Number of the **Size** field defines the number of data sets that are generated using the HYSYS model. Increasing this number increases the likelihood that the Parametric model is a "good fit" for the flowsheet model, however, the data takes longer to generate.

Choosing the Number of Data Sets

The number and range of your data sets should span the intrinsic dimension of your variable set. For example, completely span the range of your variables once all constant or linear relationships have been removed. Failure to do this may cause the Neural Network to fit a constant or linear relationship between two variables when a more complex one exists.

When the data has been generated, and the **Init/reset** button on the Training tab is pressed, the PM Utility displays a list of the MLP units, the number of inputs, the number of outputs, and the number of hidden units.

A very rough rule of thumb, used by some researchers, is to count the number of weights in the network and multiply by 10. If you have an, *n,* input network with, k, hidden units and, *m,* outputs, then the number of weights (internal parameters adjusted during training) is: $(n+1)\mathrm{k}+(k+1)\mathrm{m}$

So if n = 13, m = 12, and k = 13, there will be 350 weights in the network. In practice, one can often manage with the fewest amount of data sets, but you should not have more weights than training examples.

This number of input, output, and hidden training variables in the NN is only available once the data has been generated, and the relationships between the selected input and output variables are assessed. As such an incremental approach to generating data is recommended. Assume that there is only one hidden layer (in other words, in the above k = 1) and generate $n + 1 + 2m$ data sets (48 data sets for this example).

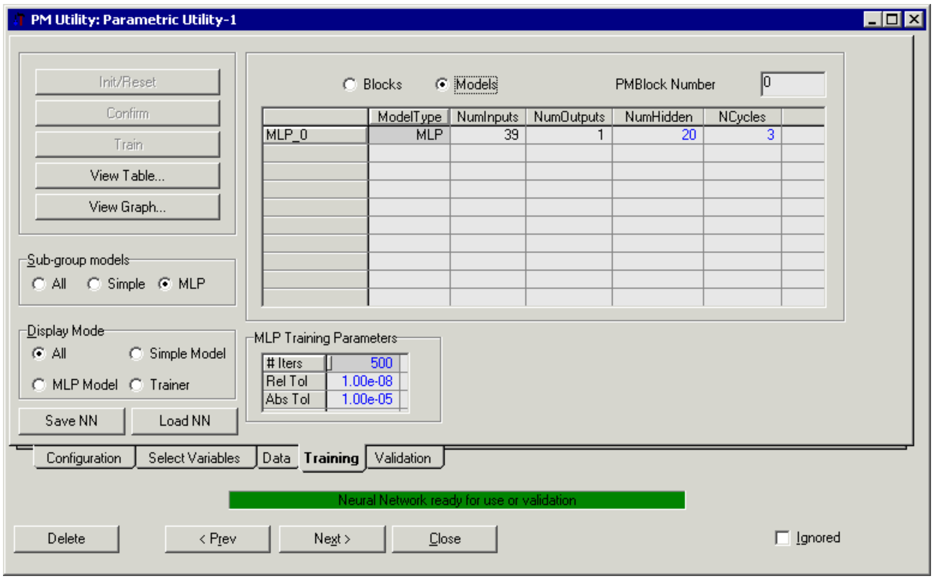
Proceed to the Training tab and press the **Init/Reset** button. The number of Input, Output, and hidden units in each MLP are displayed. Calculate the number of weights. If the number of weights are less than the number of data sets that have been generated, then you may consider training the NN. If the number of weights is greater than the number of data sets generated, return to the Data tab and click the **Append to Existing** radio button. Enter in a completely new set of data (using the same data does not help), and click **Generate Data** button again. Repeat this process until enough data sets have been generated, and appended to the original data file.

This is only a rough rule of thumb. There is no substitute for understanding the system being modelled and looking at the data to check the regions where there are rapid changes between input and output, and then providing more examples in those regions.

Training Tab

The Training tab allows you to generate a Parametric model based on the HYSYS training data. Data sets generated on the Data tab are used as training variables. The training algorithm determines the parameter values of the neural network model based on the input and output data sets. The end result is a Parametric model which approximates its HYSYS model counterpart.

Figure 14.82



The set of Training buttons are described below:

Button	Description
Init/Reset	Use this button before running the training algorithm or when you need to reset the Parametric model.
Confirm	Allows you to confirm the current training configuration.
Train	Initiates the training algorithm to train the neural network based on the data sets generated by the HYSYS model.

Button	Description
View Table...	Allows the viewing of training data in table format. Compares the HYSYS training data with Parametric model data.
View Graph...	Allows the viewing of training data in graphical format. Compares the HYSYS training data with Parametric model data.

The Sub-group models group allows filtering of neural network data using three radio buttons as described below:

Radio Button	Description
All	Displays both Simple and MLP combined.
Simple	Displays constant and linear variables. Those relationships that are not modeled by the MLP.
MLP	Displays Multilayer Perceptrons. Those relationships that are modeled by neural networks.

The Display Mode group displays model data in the table based on the radio button selected.

Radio Button	Description
All	All information associated with the model (type of model, number of inputs/outputs, and all variables associated with neural net trainer).
MLP Model	Displays the model type and information regarding NN representation parameters.
Simple Model	Displays the model type, number of inputs and outputs.
Trainer	Information that describes the training parameters for a given model.

You can also change the number of hidden layers in the MLP, and the number of cycles (in other words, number of times the data is presented to the nodes). Changing these may affect the efficiency of your model and is only recommended for the advanced user.

The matrix group and the buttons below are described in the following table:

Object	Description
Blocks radio button	Allows you to can examine the structure if data is put into multiple blocks or NN.
Models radio button	Allows you to examine the structure if data is put into model types. Model types available are: Manipulated, Constant, Ignored IntStr, and Simple Linear.
PMBlock Number field	Displays the PM block number.

In Advanced Mode, the following buttons are available on the Training tab:

Refer to **Section 14.14.4 - Neural Network (NN) Manager** for more information.

Button	Description
Load NN	Allows you to load the Neural Network (*.nn file) into another Parametric Utility (of the same configuration), same Parametric Utility, or used with the Neural Network Manager.
Save NN	Allows you to save a trained Neural Network to a specific *.nn file.

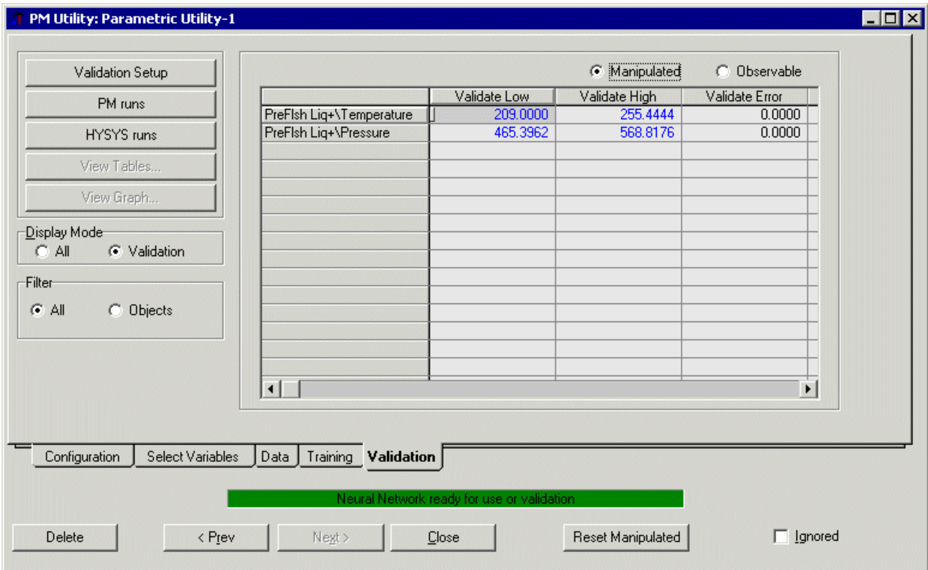
When in Advanced Mode, you can modify the following parameters:

Parameter	Description
#Iters	Maximum number of passes the training algorithm will take.
Relative Tolerance	Ratio of the change in error between two iterations to the actual error. If the value is below the tolerance training will stop.
Abs Tolerance	If the training error is below this value, training will stop.

Validation Tab

The Validation tab is the final phase in developing a Parametric model. On this tab, validation of the model is performed by generating validation points using both the Parametric model and the HYSYS model and comparing the results.

Figure 14.83



The Validation options are described below:

Button	Description
Validation Setup	Allows you to configure validation setup options which include the Number of Validation Points and the Random Speed used in generating the validation points.
PM runs	Runs the Parametric model to generate validation data based on the Parametric model.
HYSYS runs	Runs the HYSYS model to generate validation data based on the HYSYS model.
View Tables...	Allows the viewing of validation data in table format. Compares the HYSYS validation data with Parametric model data.
View Graph...	Allows the viewing of validation data in graphical format. Compares the HYSYS validation data with Parametric model data.

The Display Mode group has two radio buttons:

Radio Button	Description
All	Displays all the variable information.
Validation	Displays the validation range and error.

The Filter group filters objects based on four radio button selections:

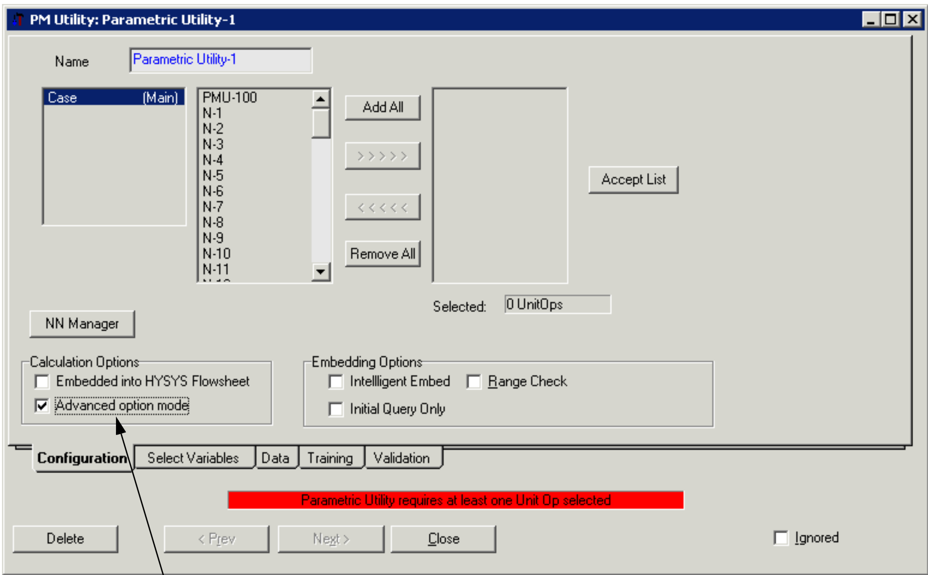
Radio Button	Description
All	Shows all variables. Objects are filtered differently depending upon whether they are manipulated or observable.
Objects	Shows the variables.filtered by the flowsheet objects.
Simple Linear	Shows the variables which have a constant or linear relationship.
MLP Models	Shows the variables which are used in the MLP model.

After you have a trained NN, you can embed it into the HYSYS flowsheet to replace the objects you selected earlier by the trained NN; use the Embedded into HYSYS Flowsheet checkbox on the Configuration tab. If your streams are over specified, HYSYS filters the stream information for you to avoid consistency errors.

14.14.4 Neural Network (NN) Manager

You can access the Neural Network Manager by clicking the NN Manager button on the Configuration tab of the Parametric Utility property view.

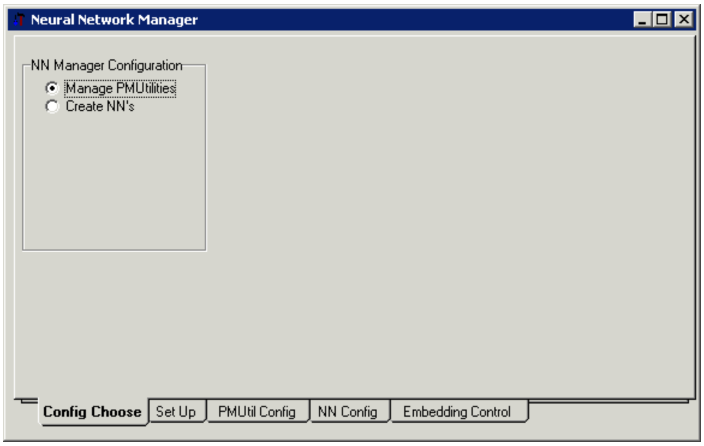
Figure 14.84



The NN Manager button only appears when you select the Advanced option mode checkbox in the Calculation Options group of the Parametric Utility Configuration tab.

The Neural Network Manager allows you to switch Neural Network (NN) Objects into appropriately configured Parametric Utilities, and to generate simple Neural Network's from external data.

Figure 14.85



The Config Choose tab allows you to switch between the two NN Manager modes:

- PMUtilities Manager
- NN Creation

PMUtilities Manager Mode

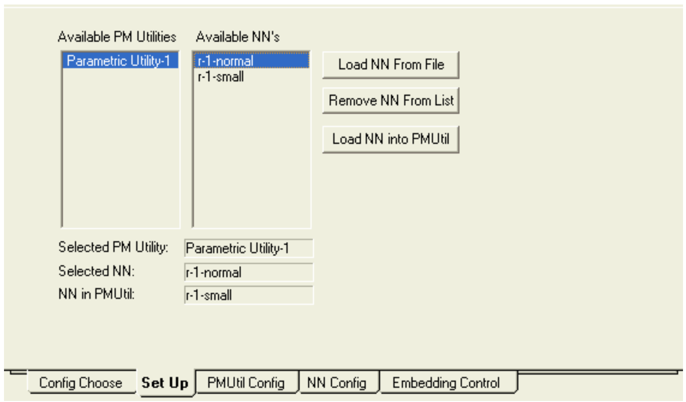
There are four tabs available for the PMUtilities Manager mode:

- Set Up
- PMUtil Config
- NN Config
- Embedding Control

Set Up Tab

The Set Up tab displays the list of available Parametric Utilities and Neural Network Objects associated with the Parametric Utilities.

Figure 14.86



The Set Up tab consists of the following fields:

Field	Description
Selected PM Utility	Displays the Parametric Utility that you have selected from the list of available PM Utilities
Selected NN	Displays the Neural Network (NN) object you have selected from the list of available NN's.
NN in PMUtil	Displays the Neural Network (NN) object that is loaded in the Parametric Utility.

The following buttons allow you to load, remove or associate a Neural Network object with the Parametric Utilities.

Button	Description
Load NN From File	Loads a Neural Network object into the Neural Network Manager and associates it with the Parametric Utility that you selected from the list of available PM Utilities.

Button	Description
Remove NN From List	Removes the selected Neural Network object from the list of available NN's and the Neural Network Manager.
Load NN into PMUtil	Loads the selected Neural Network object from the list of available NN's into the selected Parametric Utility from the list of available PM Utilities.

Parametric Utilities which have been trained are automatically added to the list of available PM Utilities.

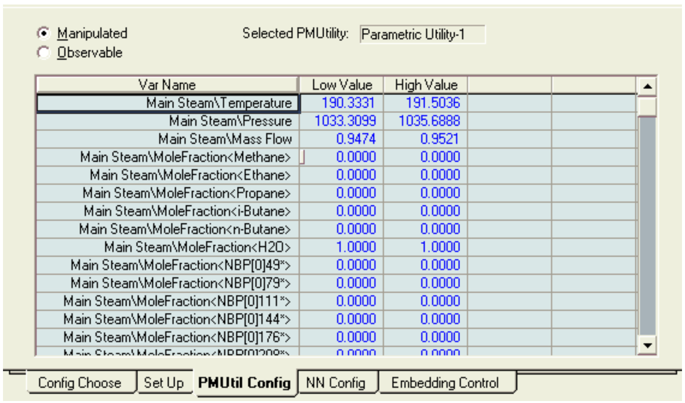
Neural Network objects have to be created from a trained Neural Network; on the Training tab of a Parametric Utility property view, in advanced mode, you can save or load a NN (.nn file) using the Save NN or Load NN buttons.

Neural Network objects must have the same configuration as the Parametric Utility object they are being loaded into. Stream and unit operation names can be different, but the list of Manipulated and Observable variables must have the same type of variable in the same order.

PMUtil Config Tab

On the PMUtil Config tab you can view a list of the Manipulated and Observable variables for the Parametric Utility you selected from the list of available PM Utilities on the Set Up tab.

Figure 14.87

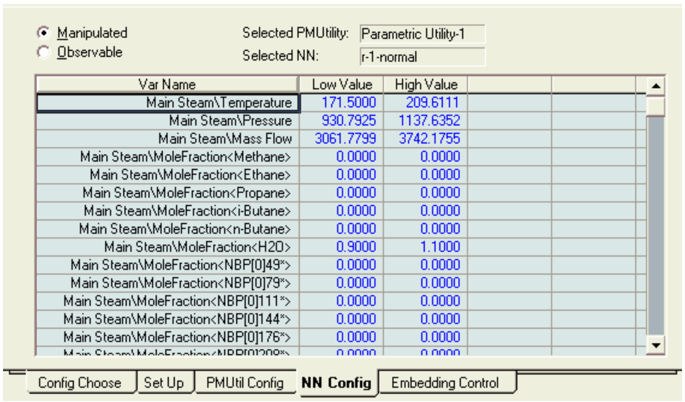


You can also view the Manipulated and Observable variables high and low values for the Parametric Utility you selected from the list of available PM Utilities on the Set Up tab.

NN Config Tab

On the NN Config tab you can view a list of the Manipulated and Observable variables for the Neural Network object you selected from the list of available NN's on the Set Up tab.

Figure 14.88

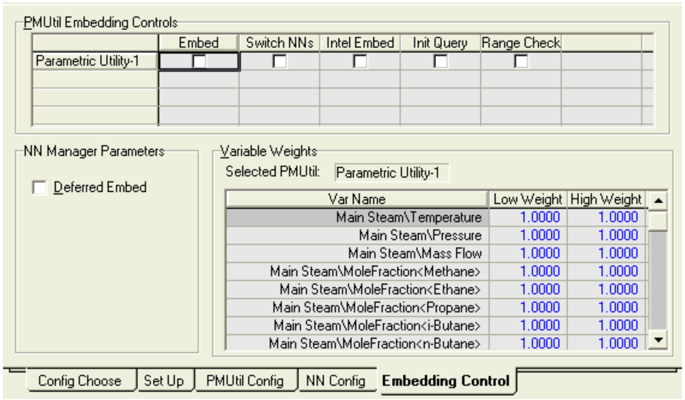


You can also view the Manipulated and Observable variables high and low values for the Neural Network object you selected from the list of available NN's on the Set Up tab.

Embedding Control Tab

The Embedding Control tab allows you to control the different embedding options.

Figure 14.89



The Embedding Control tab consists of three groups:

- PMUtil Embedding Controls are described in the table below:

Controls	Description
Embed	Embeds the Parametric Utility into the flowsheet.
Switch NN's	Switches the Neural Network's into the Parametric Utility automatically to the one that fits the manipulated variable values most appropriately.
Intel Embed	The PMUtil makes choices to maximize a successful embedding, if they need to be made.
Init Query	If the you need to be queried on an embed, you will only be queried once.
Range Check	If the manipulated variable values are outside the range than the Neural Network in the Parametric Utility was trained upon, then the Parametric Utility is unembedded.

- **NN Manager Parameters**. You can select the **Deferred Embed** checkbox, if the Neural Network Manager needs to switch a Neural Network into a Parametric Utility, but there is an adjust or recycle in the scope of the Parametric Utility, then the switch will be done after the adjusts and recycle have converged.

- **Variable Weights**. The weights used to determine the NN which fits the manipulated variable values most appropriately. The higher the values of the weight, the greater effect that variable bound has on selecting the NN. A weight of 0 shows the variable bound will have no effect.

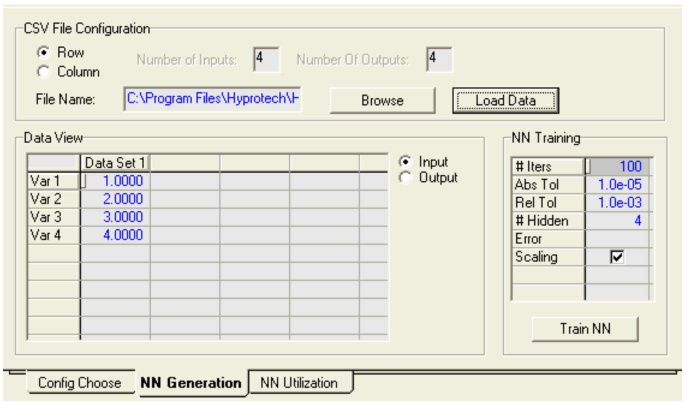
NN Creation Mode

There are two tabs available for the NN Creation mode:

- NN Generation
- NN Utilization

NN Generation Tab

Figure 14.90



There are three groups on the NN Generation tab:

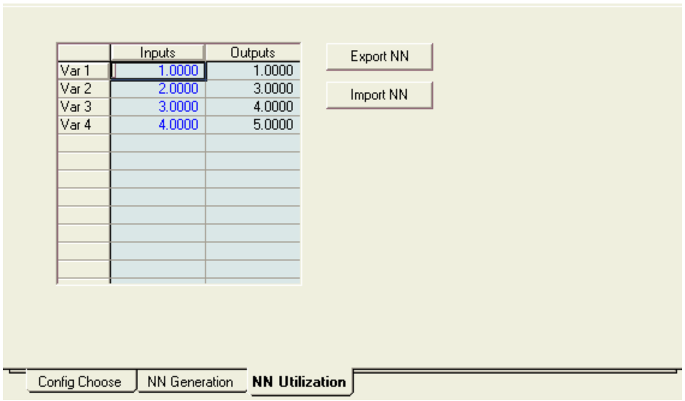
Refer to **Data File Format Group** section from **Section 5.6.2 - Design Tab** in the **HYSYS User Guide**, for more information on specifying the row or column format for a data file.

- **CSV File Configuration**. Allows you to specify the row or column format for the data file, browse for the data file, and load the data file into the Neural Network Manager by clicking the **Load Data** button.
- **Data View**. Displays the data loaded from the CSV file.
- **NN Training**. Allows you to modify training parameters (#Iterations, Abs Tolerance, Relative Tolerance, #Hidden Layers, Scaling), and displays the Error of the trained Neural Network. When you click the **Train NN** button, it starts the training algorithm.

NN Utilization Tab

The NN Utilization tab allows you to save or load trained Neural Network's (.mlp files), and utilizes the *.mlp files for displaying the Inputs and Outputs in the table.

Figure 14.91



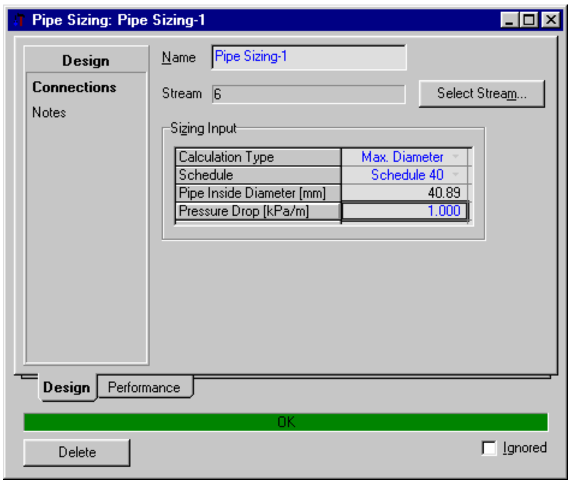
The Neural Network's stored here are in a different format from those used in the PMUtil Manager mode as they do not contain variable type information. The two formats are currently not compatible.

14.15 Pipe Sizing

To add the Pipe Sizing utility, refer to the section on **Adding a Utility**.

With the Pipe Sizing utility you can perform design calculations on any of the case streams. Results include pipe schedule, pipe diameter, Reynolds number, friction factor, and so forth.

Figure 14.92



14.15.1 Design Tab

The Design tab contains the following pages:

- Connections
- Notes

Connections Page

On the Connections page, you can select the stream that represents the pipe you want to size, the calculation type and characteristics of the pipe.

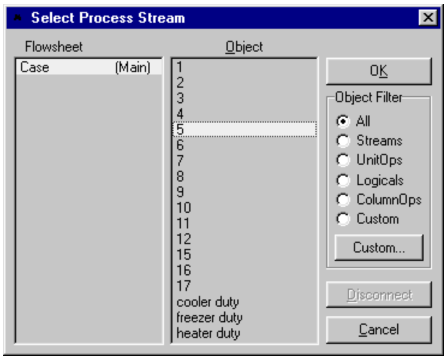
You can also remove or change the streams to be used by clicking the Select Stream button, and changing the selection in the Select Process Stream property view.

General Procedure

The following are the steps to pipe size a selected stream.

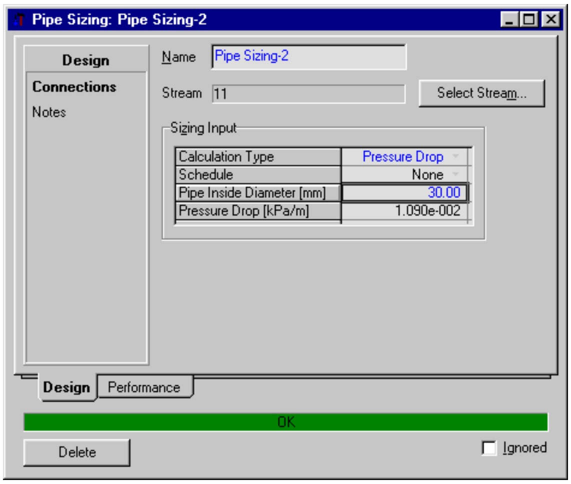
1. On the **Design** tab, click the **Connections** page.
2. Click the **Select Stream** button. The Select Process Stream property view appears.

Figure 14.93



3. Select a stream for the analysis from the Object list.
4. Click the **OK** button to return to the Pipe Sizing property view.
5. You must select a calculation type from the **Calculation Type** drop-down list. The options available are:
   - **Max. Diameter**. The input required includes the pipe schedule, and the pressure drop in the pipe.
   - **Pressure Drop**. The input required includes the pipe schedule, and the pipe diameter.

Figure 14.94



The following fields are available for each stream chosen.

Object	Description
Calculation Type	Allows you to choose between two calculation types: • Max. Diameter • Pressure Drop
Schedule	Allows you to select a pipe schedule. You have four choices: • None • Schedule 40 • Schedule 80 • Schedule 160 If you selected Pressure Drop as your calculation type, the pipe schedule is automatically set as None.
Diameter	If you selected Pressure Drop as your calculation type, then you have to enter a value for the pipe's actual inner diameter. HYSYS then calculates the pressure drop.
Pressure Drop	If you selected Max. Diameter as your calculation type, then you have to enter a value for the pressure drop. HYSYS then calculates the pipe's actual inner diameter.

Notes Page

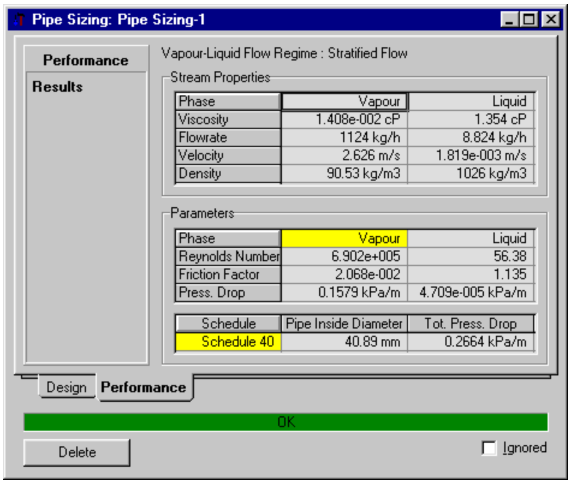
For more information, refer to **Section 1.3.5 - Notes Page/Tab**.

The Notes page provides a text editor, where you can record any comments or information regarding the utility, or to your simulation case in general.

14.15.2 Performance Tab

You can examine the results of the pipe sizing utility on the Results page of the Performance tab.

Figure 14.95



14.16 Production Allocation Utility

To add the Property Balance utility, refer to the section on **Adding a Utility**.

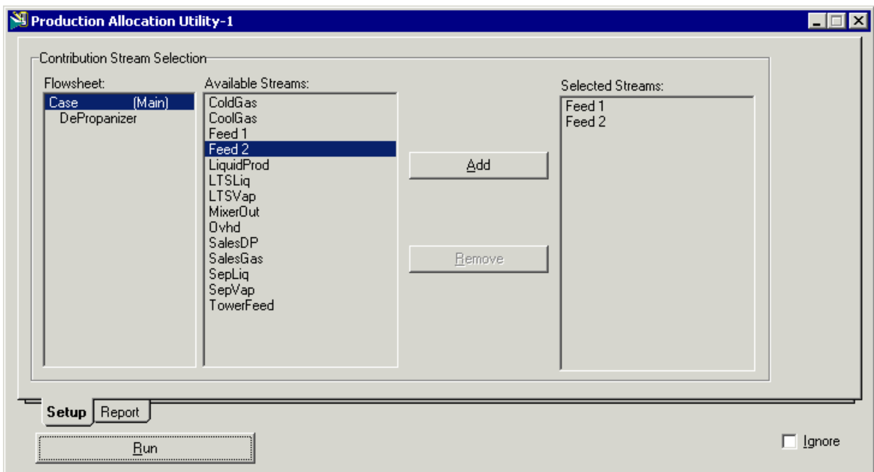
The Production Allocation Utility enables you to track the contribution of selected streams to other down flowsheet streams. The contribution is tracked on a compositional flow or percentage basis.

Use of the Production Allocation utility is particularly relevant in scenarios where a model depicts a system that relies on multiple suppliers for inlet feeds and you want to track the individual supplier contributions to the resulting products.

The utility does not navigate into Column Subflowsheets and does not support the use of reactions or reactors.

Black Oil streams must first be translated in order to be used with the utility.

Figure 14.96



14.16.1 Setup Tab

The Setup tab enables you to select the flowsheet and streams within the flowsheet. Typically feed streams are selected.

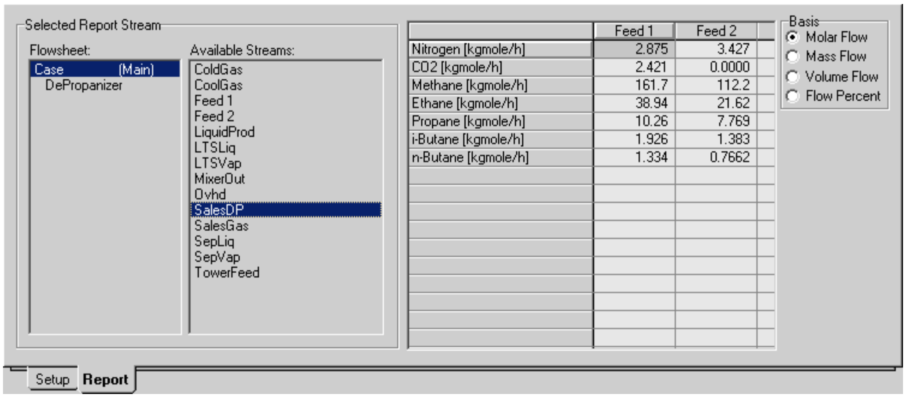
Object	Description
Flowsheet list	Enables you to select the flowsheet containing the streams you want to track.
Available Streams list	Enables you to select the streams available in the selected flowsheet.
Selected Streams list	Displays the list of streams you have added into the Production Allocation utility.
Add button	Enables you to add the selected stream (in the Available Streams list) into the Production Allocation utility for tracking.
Remove button	Enables you to remove the selected stream (in the Selected Streams list) from the Production Allocation utility.
Run button	Enables you to track the composition flow and percentage basis of the added streams. The results are displayed in the **Report** tab.

Every time a variable or property of the streams are changed, the Run button needs to be clicked to get the new values/results.

14.16.2 Report Tab

The Report tab enables you to view the component flow rate of the selected streams.

Figure 14.97



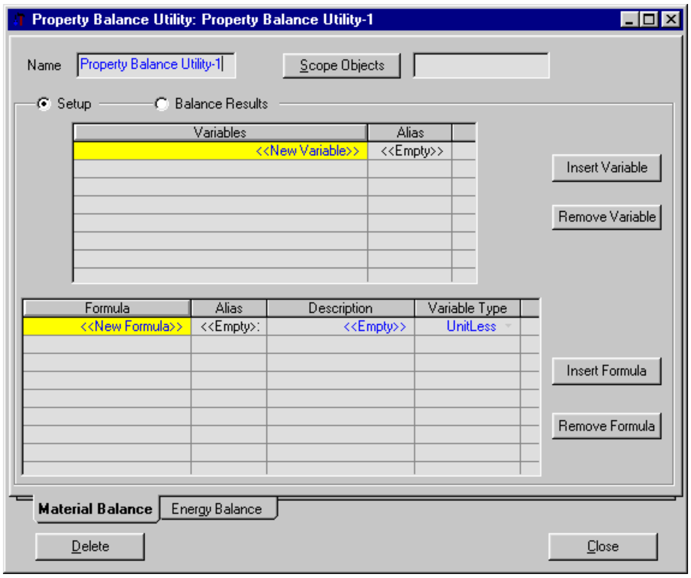
Object	Description
Flowsheet list	Enables you to select the flowsheet containing the stream you want to view.
Available Streams list	Enables you to select a stream available in the selected flowsheet.
Selected Report Stream table	Displays the contribution (component or percent flow rate) from the added streams to the selected stream.
Basis group	Contains radio buttons that enables you to select the basis of the contribution flow rate. The types of basis available are: Molar, Mass, Volume, and Flow Percent.

14.17 Property Balance Utility

To add the Property Balance utility, refer to the section on **Adding a Utility**.

The Property Balance Utility allows you to inspect material and energy balances across the entire flowsheet or across selected operations. The figure below shows the Property Balance utility property view, when you first generate the utility.

Figure 14.98



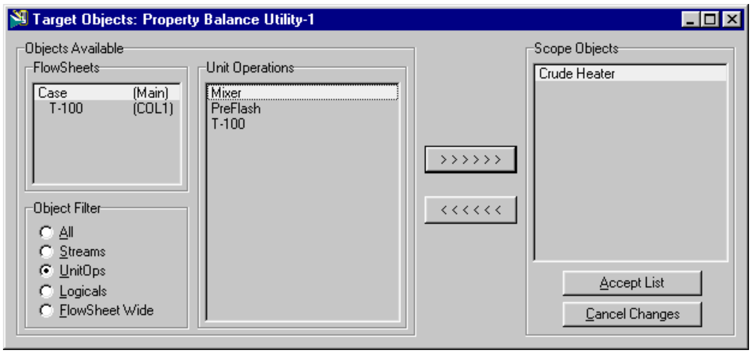
The Property Balance utility property view contains a Name field, a Scope Object button, a Delete button, a Close button, and two tabs: Material Balance and Energy Balance.

You can change the name of the Property Balance utility by clicking the Name field and entering a new name.

Selecting Scope for Balance Calculations

Click the Scope Object button to select what operations you want to perform your material or energy balance calculation over. The Target Objects property view appears.

Figure 14.99



The Target Objects property view contains two groups and three sub groups.

The Objects Available group contains tools to help you select the objects you want to perform the balance calculation over.

Generalized Procedure for Selecting Objects

1. Select the flowsheet containing the objects you want from the FlowSheets group. Use the Object Filter radio buttons to help narrow your search.
2. Select the objects you want from the object list in the Objects Available group.

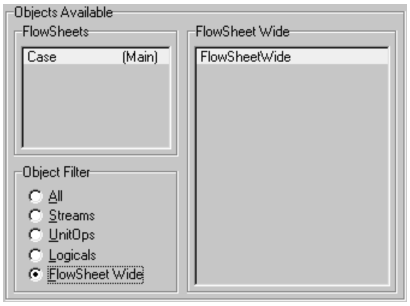
   Use the **SHIFT** or **CTRL** keys to select more than one object from the list.
3. Click the 

 button to move the selected objects from the Objects Available group to the Scope Objects group.

You can remove objects from the Scope Objects group by selecting the objects you do not want from the list, and clicking the <<<<<< button.

To perform a material or energy balance over the entire flowsheet, you have to select all the operations in the flowsheet. You can also perform a material or energy balance over the entire main flowsheet by selecting the FlowSheet Wide radio button in the Object Filter group, and adding the FlowSheet Wide variable from the FlowSheet Wide group.

Figure 14.100

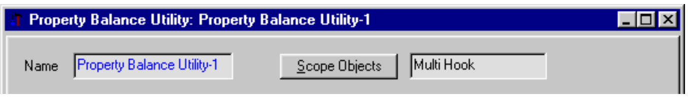


4. Once you have added all the objects you want in the Scope Objects group, click the **Accept List** button.

   Click the **Cancel Changes** button to exit the Target Objects property view without making any changes.

5. After clicking the **Accept List** or **Cancel Changes** button, you return to the Property Balance utility property view.
6. If you selected one object for balance calculation, the field beside the **Scope Objects** button displays the name of the object. If you had selected more than one object, the field displays the words **Multi Hook**. This indicates that more than one object was selected.

Figure 14.101



14.17.1 Material Balance Tab

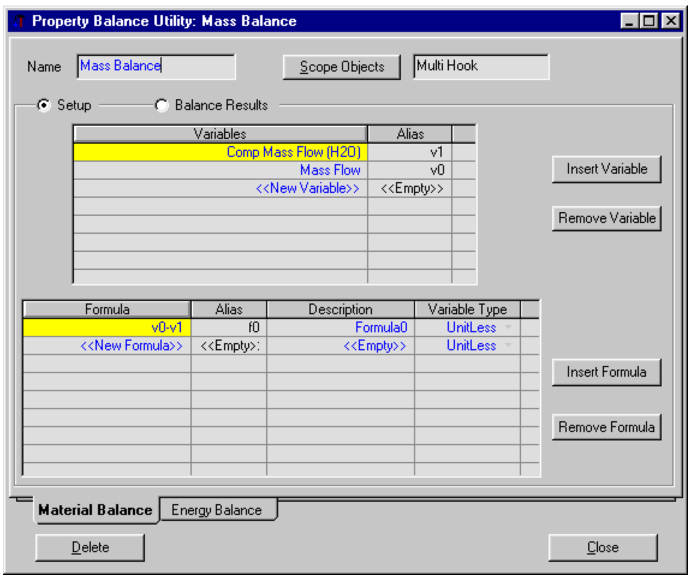
The Material Balance Tab contains two radio buttons that control the property view in the tab:

- Setup
- Balance Results

Setup Radio Button

The utility generates a balance on any stream property or combination of stream properties you select. You can set up multiple balances and view results of each balance in turn. On the Setup property view, you define the stream properties of interest (the variables), and the relationships between them (the formulas).

Figure 14.102



The following table contains the description of the objects in the Setup property view.

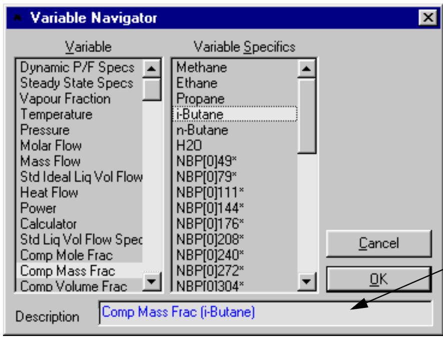
Objects	Description
Insert Variable	Allows you to insert a variable to the balance calculation. The variable you selected appears in the Variable column.
Remove Variable	Allows you to remove a variable from the table, select the variable you want to remove and click this button
Variable column	Displays the name of the variable you added to the balance calculation. The name is based off the text entered in the Description field from the Variable Navigator property view.
Alias column	Displays the name designated to the variable by HYSYS. The designated name is used to represent the variable when entering a formula equation.
Insert Formula	Allows you to insert a space for a formula in the formula table.
Remove Formula	Allows you to remove a formula from the table, select the formula you want to delete and click this button.
Formula column	Allows you to enter a formula equation.
Alias column	The column displays the name designated to the created formula by HYSYS. The designated name is used to represent the formula when entering a formula equation.
Description column	Allows you to enter a description about the formula.
Variable Type column	You can select the variable unit type for the calculated result of the formula in this column. The variable types available are contained in the drop-down list. HYSYS automatically assigns variable unit types to simple formula. An example is a formula with a temperature variable being multiplied by 2, HYSYS assigns Temperature as the formula's variable unit type.

Adding a Variable

To add a variable:

1. Click the **Insert Variable** button. The Variable Navigator property view appears.

Figure 14.103



If you want, you can enter a different description for the selected variable in the Description field.

You can also insert a variable by double-clicking the **<<New Variable>>** cell.

You can also edit the variable selected by double-clicking on the variable cell you want to change.

2. Select the variable you want from the Variable list, and Variable Specifics list if there are any.

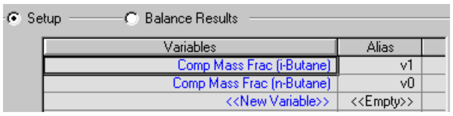
   You cannot add the following variables from the Variable list:

   - Heat Flow
   - Molar Enthalpy
   - Mass Enthalpy
   - Molar Entropy
   - Mass Entropy
   - Phase Enthalpy
   - Phase Entropy
   - Phase Heat Flow

3. Click the **OK** button when you are done selecting a variable.

4. You return to the Property Balance utility property view. Notice that the variable you added appears in the variable table.

Figure 14.104

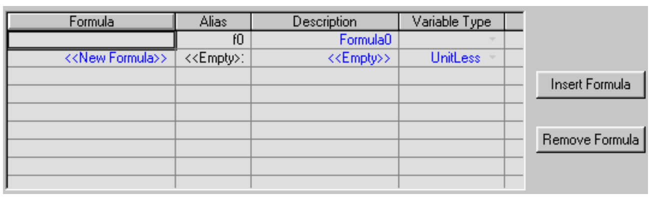


Adding a Formula

To add a formula:

1. Click the **Insert Formula** button, an empty row for the formula appears in the formula table.

Figure 14.105

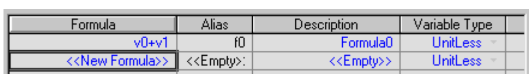


You can also enter a new formula by entering the formula in the <<New Formula>> cell.

The Insert Formula button can be used to insert a formula in between a list of formulas.

2. Enter a formula equation in the Formula cell. To manipulate any of the variables from the variable list, you have to use the names of the variables from the Alias column.

Figure 14.106



The formulae follow the same syntax as the spreadsheet.

To view the syntax available in HYSYS, add a spreadsheet to the PFD, open the spreadsheet property view, and click the **Function Help** button.

3. You can change the Description of the formula by entering a new description in the Description cell. HYSYS sets a default description of the word **Formula** and the number designated to the formula. You can also select a variable type for the formula results. HYSYS default selection for the variable type is **Unitless**.

   You can also enter a formula containing another formula. An example is the table containing both formulas, f0 = v0-1 and f1 = f0*10. So formula f1 contains another formula f0.

Figure 14.107

Formula	Alias	Description	Variable Type
v0-1	f0	Formula0	Temperature
f0*10	f1	Formula1	UnitLess
<<New Formula>>	<<Empty>>	<<Empty>>	UnitLess

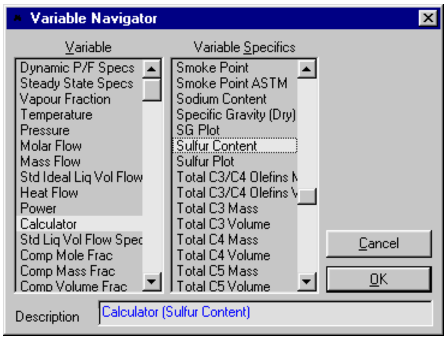
Setting up a Property Balance

Suppose you want to look at the sulfur balance across a group of objects, and you are using refinery assays. You can do the following:

1. Add a Mass Flow variable (v0).
2. Add a Sulfur Content variable (v1).

   This variable is located by selecting **Calculator** in the Variable list and **Sulfur Content** in the Variable Specifics list from the Variable Navigator property view.

Figure 14.108

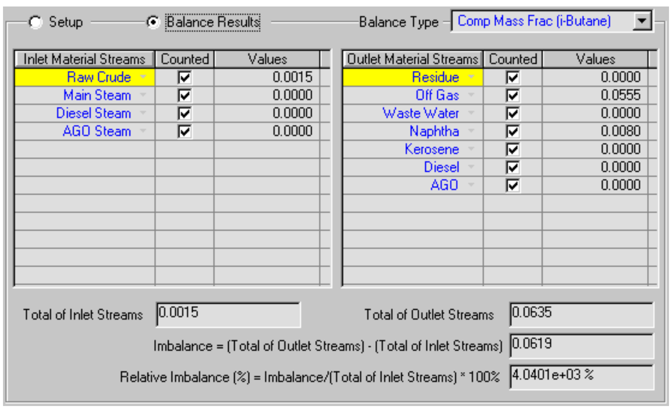


3. Add a formula of **v0*v1/100** to get the mass flow of sulfur.
4. View the results of this formula by selecting the **Balance Results** radio button, and selecting the formula from the Balance Type drop-down list.

Balance Results Radio Button

You can view the results of the selected variables and formula by selecting the Balance Results radio button.

Figure 14.109



Refer to **Section 12.2.3 - Dynamics Tab** for more information regarding Feed Block and Product Block streams.

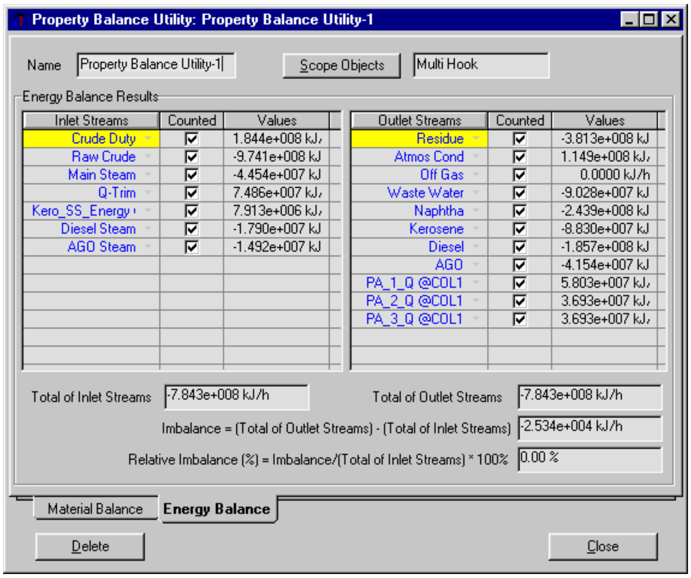
The following table contains the description of each object on the tab when you select the Balance Results radio button.

Object	Description
Balance Type	You can select the type of balance you want to see from the drop-down list. The type of balance comes from the variables you added and formulas you created when the Setup radio button was selected.
Inlet Material Stream column	Displays all the inlet streams considered for the balance calculation. If you selected one operation, then the inlet streams for that particular operation is considered. If you selected all the operations in the flowsheet, only the Feed Block streams are considered.
Outlet Material Stream column	Displays all the outlet streams considered for the balance calculation. If you selected one operation, then the outlet streams for that particular operation is considered. If you selected all the operations in the flowsheet, only the Product Block streams are considered.
Counted column	You can indicate which stream you do not want in the balance calculation by clearing the checkbox of the associated stream in this column. The default setting for the checkbox is active.
Values column	Displays the value of the variable for the associated stream. The variable is the selected variable from the Balance Type drop-down list.
Total of Inlet Stream	Displays the sum of the values in the Value column for the inlet streams.
Total of Outlet Stream	Displays the sum of the values in the Value column for the outlet streams.
Imbalance	Displays the result of subtracting the total inlet stream value from the total outlet stream value.
Relative Imbalance	Displays the percentage result of dividing the imbalance value by the total inlet stream value and multiplying that value by 100.

14.17.2 Energy Balance Tab

The Energy Balance tab displays the energy balance results across the operations you selected in the Targets Object property view.

Figure 14.110



The table below contains the description of the objects on the Energy Balance tab.

Refer to **Section 12.2.3 - Dynamics Tab** for more information regarding Feed Block and Product Block streams.

Object	Description
Inlet Stream column	Displays all the inlet streams considered for the balance calculation. If you selected one operation, then the inlet streams for that particular operation is considered. If you selected all the operations in the flowsheet, only the Feed Block streams are considered.
Outlet Stream column	Displays all the outlet streams considered for the balance calculation. If you selected one operation, then the outlet streams for that particular operation is considered. If you selected all the operations in the flowsheet, only the Product Block streams are considered.
Counted column	You can indicate which stream you do not want in the balance calculation by clearing the checkbox of the associated stream in this column. The default setting for the checkbox of all the stream is active.

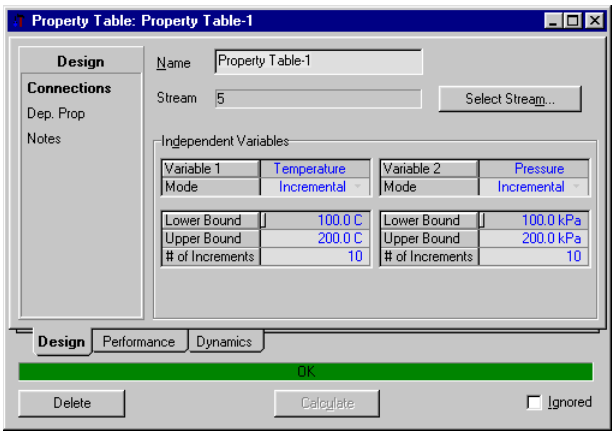
Object	Description
Values column	Displays the heat flow value of the associated stream.
Total of Inlet Stream	Displays the sum of the values in the Value column for the inlet streams.
Total of Outlet Stream	Displays the sum of the values in the Value column for the outlet streams.
Imbalance	Displays the result of subtracting the total inlet stream value from the total outlet stream value.
Relative Imbalance	Displays the percentage result of dividing the imbalance value by the total inlet stream value and multiplying that value by 100.

14.18 Property Table

To add the Property Table utility, refer to the section on **Adding a Utility**.

The Property Table utility allows you to examine property trends over a range of conditions in both tabular and graphical formats. Using a stream of known composition, you can target two independent variables and their respective ranges of interest. The range of each independent variable is distinct, and can be set as either an incremental range or a selection of specific values. Next, you can relate which dependent variables are to be displayed at each combination of the independent variables.

Figure 14.111



14.18.1 Design Tab

The Design tab contains the following pages:

- Connections
- Dep. Prop
- Notes

Connections Page

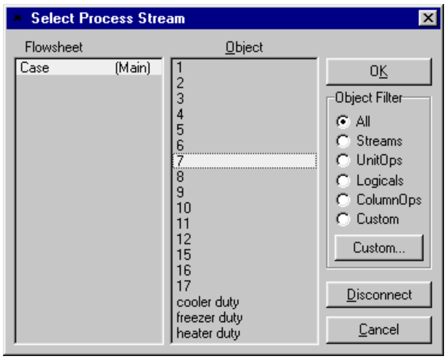
On the Connections page, you can select the stream, the two independent properties, the range of values, and increments you want the utility to display.

Independent Variables

The following are the general steps required to set the independent variables.

1. On the **Design** page, click the **Connections** page.
2. Click the **Select Stream** button, and the Select Process Stream property view appears.

Figure 14.112



3. Select a stream for the analysis from the Object list.
4. Click the **OK** button to return to the **Connections** page.

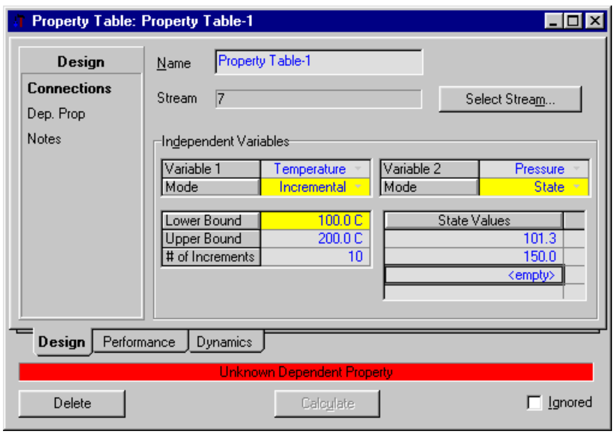
5. On the **Connections** page, identify one or two independent variables in the Variable 1 and Variable 2 (if desired) input cells. The options include:
   - Pressure
   - Temperature
   - Vapour Fraction
   - Enthalpy
   - Entropy

One of the independent variables must be either Pressure or Temperature. If the first variable selected is not Temperature or Pressure, the drop-down list for the second variable can be limited to Temperature, Pressure and Not Set.

6. Next, you can select the **Mode** for the independent variable(s). There are two options:
   - **Incremental**. The input required includes the number of increments, and values for the upper and lower bounds. The dependent variable(s) are calculated at each increment within the range.
   - **State**. You can input an unlimited number of specific values for the independent variable in the State Values matrix.

For the incremental variable(s), specify an upper bound, a lower bound, and the number of increments.

Figure 14.113



Next, you need to specify the dependent property as stated in the red status bar.

Dep. Prop Page

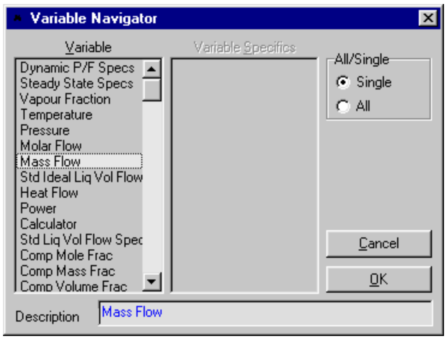
On the Dep. Prop page, you can select the dependent variable.

Dependent Variables

The following are the general steps required to set the dependent variables.

1. On the **Design** page, click the **Dep. Prop** page.
2. Click the **Add** button, and the Variable Navigator property view appears.

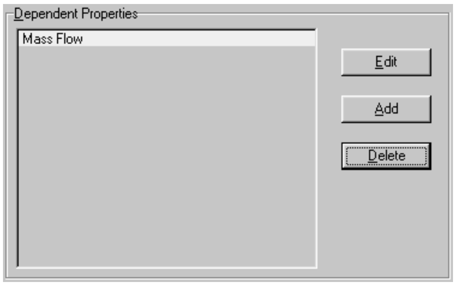
Figure 14.114



3. Select a stream for the analysis from the Object list.

4. Click the **OK** button to return to the **Dep. Prop** page.

Figure 14.115

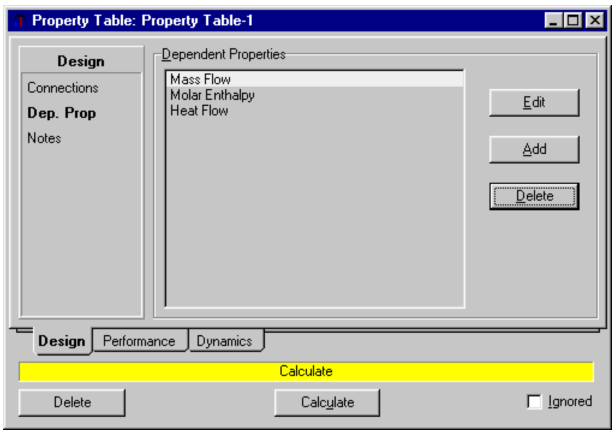


You can change the dependent variable by selecting the variable from the list and clicking the Edit button.

You can remove the dependent variable from the list by selecting the variable, and clicking the Delete button.

5. Repeat steps #2 to #4 to add more dependent variables.
6. Click the **Calculate** button once you've added all your variables.

Figure 14.116



The **Calculate** button is located at the bottom of the property view. The **Calculate** button is only available when:

- HYSYS has not calculated the values for the Property Table.
- You have selected a stream, independent variable and dependent variable.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page / Tab**.

The Notes page provides a text editor, where you can record any comments or information regarding the utility, or to your simulation case in general.

14.18.2 Performance Tab

You can examine the results of the property table utility in the pages on the Performance tab. The Performance tab contains the following pages:

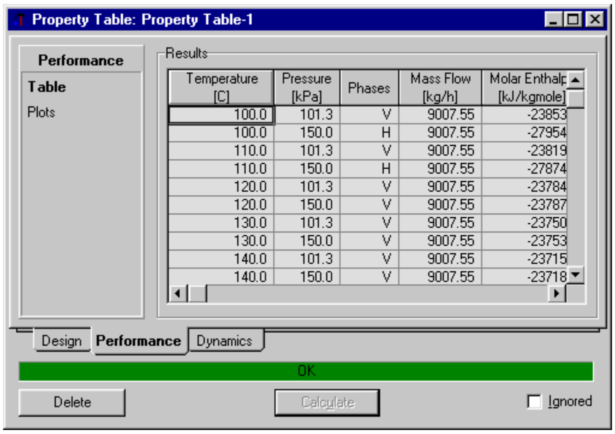
- Table
- Plots

Table Page

A table listing the results of the property table calculations can be viewed on the Table page. This page lists the independent variables, the dependent variables, and the phases present at the given conditions.

The phase column indicates the phases, which have been detected at each pair of independent property values. The V indicates vapour, L indicates a light liquid (hydrocarbon rich) phase, and H indicates the presence of a heavy liquid (aqueous) phase.

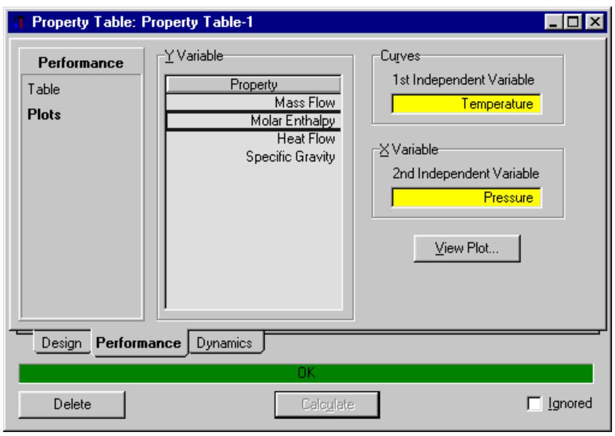
Figure 14.117



Plots Page

The Plots page allows you to display the results of the Property Table utility calculations in a graphical format.

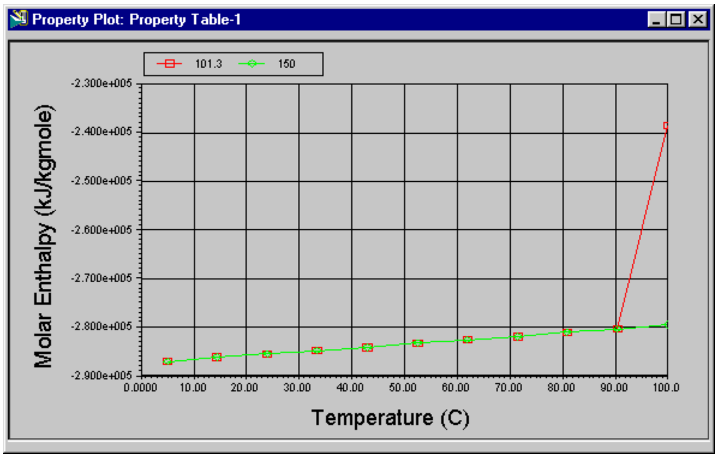
Figure 14.118



You can select which dependent variable you want to display on the y-variable by selecting it in the Y Variable group.

Click the **View Plot** button to display the plot.

Figure 14.119

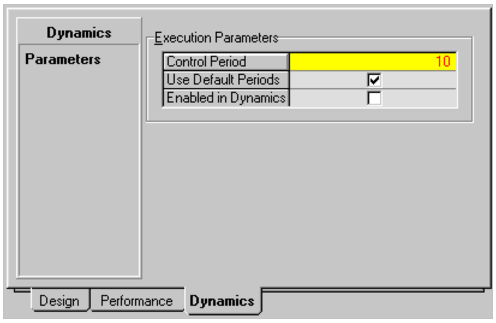


To make changes to the plot appearance, right-click in the plot area and select **Graph Control** command from the object inspect menu to access the **Graph Control Property View**.

14.18.3 Dynamics Tab

The Dynamics tab allows you to control how often the utility gets calculated when running in Dynamic mode.

Figure 14.120



The Control Period field is used to specify the frequency that the utility is calculated. A value of 10 indicates that the utility be recalculated every 10th pressure flow step. This can help speed up your dynamic simulation since utilities can require some time to calculate.

The **Use Default Periods** checkbox allows you to set the control period of one utility to equal the control period of any other utilities that you have in the simulation. For example, if you have five utilities and require them all to have a control period of 5 and currently the value is 8, with this checkbox selected if you change the value in one utility all the other utilities change. Alternatively, if you want all the utilities to have different values you would clear this checkbox.

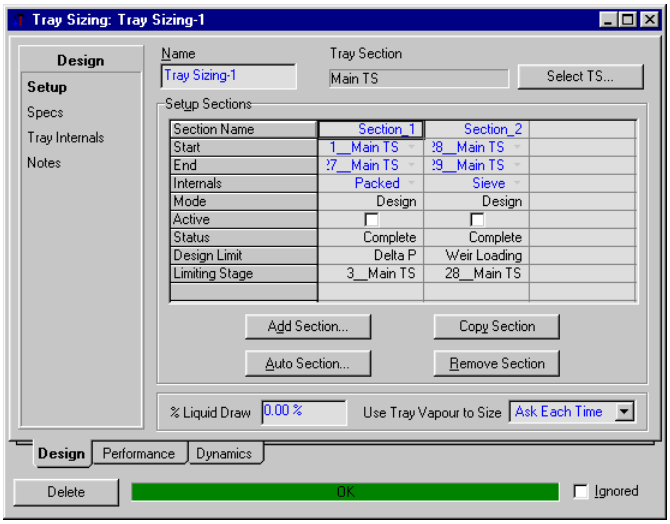
The **Enable in Dynamics** checkbox is used to activate this feature for use in Dynamic mode.

14.19 Tray Sizing

To add the Tray Sizing utility, refer to the section on **Adding a Utility**.

With the Tray Sizing utility you can perform design and rating sizing calculations on part or all of a converged column. Packing or tray information can be specified relating to specific tower internals such as tray dimensions or packing sizes, design flooding, and pressure drop specifications. Results include tower diameter, pressure drop, flooding, tray dimensions, and so forth.

Figure 14.121



The Tray Sizing utility is only available for columns with vapour-liquid flows. So this utility cannot be used to size the Liquid-Liquid Extractor.

The Tray Sizing utility must correspond to a single column flowsheet tray section.

You can set the default parameters for the Tray Sizing utility from the Session Preferences property view (from the Tools menu, select Preferences). On the Tray Sizing tab, the defaults for auto section parameters, trayed section, and packed section setups can be set.

14.19.1 Design Tab

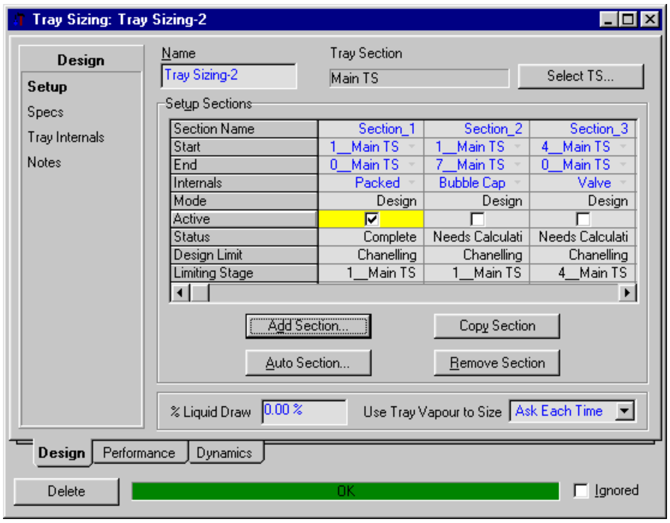
The Design tab contains the following pages:

- **Setup**. Manages the column sizing sections.
- **Specs**. Calculation mode and common tower sizing parameters.
- **Tray Interval**. Detailed internal specifications.
- **Notes**. A text editor within the utility for you to enter notes.

Setup Page

On the Setup page, you can select which column you want the tray sizing utility to calculate. This page contains several fields and a group.

Figure 14.122



You can change the name of the utility on the Setup page, by entering a new name in the Name field.

HYSYS allows you to create multiple stage sections so that you can compare column configurations with different internal types.

Therefore, a given span of tray section stages can be sized more than once within a single Tray Sizing utility. However, a give stage cannot be included in more than one active section.

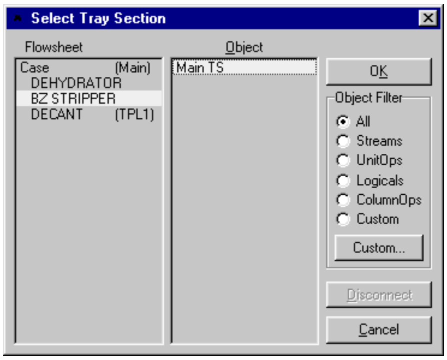
> **You can make a section active by selecting the Active checkbox for the selected section.**

Selecting the Column Trays

To select the column trays for sizing:

1. On the **Design** tab, click the **Setup** page.
2. Click the **Select TS** button. The Select Tray Section property view appears.

Figure 14.123



3. Select the column and trays from the Flowsheet list and Object list.
4. Click the **OK** button. You return to the **Setup** page.

Next, the tray section for which the sizing is desired has to be specified before starting any calculations. HYSYS automatically groups the column trays into sections, or you can add and define your own sections for the column.

Setup Sections Group

The Setup Sections group contains the options you need to generate tray sections. There is a table and four buttons in the group.

The table contains options for you to use in manipulating the tray section and the calculation method.

The four buttons at the bottom of the table allow you to manipulate the number of tray sections attached to the utility.

The buttons on the Setup page remain greyed-out until a tray section is attached to the utility.

The following table contains a description of each option in the Setup Sections table.

Row	Description
Section Name	Displays the name HYSYS designates to each tray section generated. You can change the tray section name in this field.
Start	Displays the starting stage of the tray section. You can change the tray/stage using the drop-down list.
End	Displays the stage where the tray section ends. You can change the tray/stage using the drop-down list.
Internals	You can specify each tray section's internal type. There are four groups: • Sieve • Valve • Packed • Bubble Cap

Row	Description
	Packed Sections Packed towers are calculated using either Robbins or Sherwood-Leva-Eckert design correlations for predicting pressure drop and liquid hold-up. The tower internals can be selected on the Specs and Tray Internals pages of the utility property view. You are able to specify the packing type and other parameters specifically related to the packed tower calculations on these pages, as well. **Trayed Sections** The trayed column internals are defined as sieve, valve or bubble cap. Some tray configuration parameters are common to all tray types. There is also a unique set of parameters for each individual tray type. The tower internals can be selected on the Tray Internals page of the utility property view. The calculation methods for the different trays are defined below: • Valve tray calculations are based on the Glitsch, Koch and Nutter valve tray design manuals. • Sieve tray calculations are based on the valve tray manuals for tray layout, and Mass-Transfer Operations by Treybal, (McGraw-Hill) for pressure drop, weeping, and entrainment calculations. • Bubble tray calculations are based on the method described in Design of Equilibrium Stage Processes by Bufford D. Smith, (Wiley & Sons).
Mode	The tray sizing utility has two calculation modes: • **Design**. In Design mode, HYSYS allows you to perform a design sizing based on the vapour and liquid traffic in the tower. Available design specifications for trayed and packed sections include the type of tower internals, maximum allowable pressure drop, and maximum allowable flooding. For trayed sections the maximum allowable downcomer backup, maximum allowable weir loading, and various other tray parameters can also be specified. • **Rating**. In Rating mode, HYSYS allows you to perform rating calculations based on a specified tower diameter and fixed tray configuration. If desired, some of the tray dimensions can be left unspecified and HYSYS automatically calculate design values for them. To perform a rating on a packed section, only the tower diameter is required. You can only change the **Calculation Mode** on the Specs page. On the Setup page, the mode is view-only.

Row	Description
Active	When this checkbox is selected for the tray section, the values calculated in the tray sizing utility are used in the actual column calculations. More than one section can be active in a tray sizing utility. However, the same stage cannot be included in more than one active section. Before updating the column flowsheet with the information from the tray sizing utility, you must change the default arrangement of the pressure profile information in the Column subflowsheet. 1. In the Column Runner property view, on the **Parameters** tab, click the **Profiles** page. 2. The top and bottom stage pressures must be specified instead of the condenser and reboiler pressures. The condenser and reboiler delta P specifications do not need to be changed. 3. Run the column and then return to the utility. 4. In the Tray Sizing Utility property view, on the **Design** tab, click the **Setup** page. 5. Activate those calculated column sections that you want to use in your simulation. 6. Proceed to the **Performance** tab, and click on the **Results** page. 7. Click the **Export Pressures** button, which export the pressure information to the column runner. If your column is in a recycle, there is no automatic update of the pressure profile, it must be done manually.
Status	Indicates the status of the tray sizing calculation. The status read either Complete or Incomplete on a section by section basis.
Design Limit	Indicates the design specification that was the last to be satisfied. The five design specifications are: • Minimum diameter • Pressure drop • Flooding • Weir loading (trayed sections only) • Downcomer backup (trayed sections only) This is the critical design specification that is closest to being exceeded if the tower is sized any smaller. For trayed sections, HYSYS uses individual design limits for the required active area and the required downcomer area design calculations.
Limiting Stage	The stage in the sizing section on which the design hinges. It is the stage that is closest to exceeding the design specifications. For trayed towers, there are two limiting trays; one for the tray that was closest to exceeding the design specification while satisfying the section's active area needs, and another one for satisfying the downcomer area.

The table below describes the function of the four buttons in the Setup Sections group.

Refer to the section on **Adding Tray Sections** for more information.

Refer to the section on **Copying a Tray Section** for more information.

Refer to the section on **Using Auto Section** for more information.

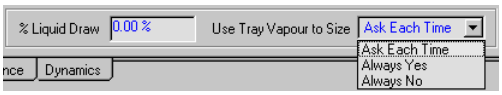
Refer to the section on **Removing a Tray Section** for more information.

Button	Description
Add Section	Allows you to add a new tray sizing section.
Copy Section	Allows you to copy any tray section already created.
Auto Section	Automatically calculates the sections in your column.
Remove Section	Allows you to delete the selected tray section. HYSYS does not ask for confirmation before removing the section.

At the bottom of the page there are two fields:

- % Liquid Draw
- Use Tray Vapour to Size

Figure 14.124

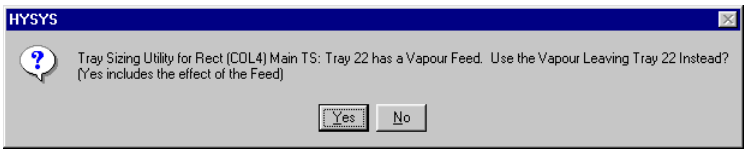


The % Liquid Draw field allows you to specify the percentage of side liquid draws to be used in the tray sizing calculations. When you specify a liquid percent HYSYS assumes that draw percentage is sitting on the tray. The default value of 0% means that no additional liquid is assumed to be on the tray. So if you enter 100%, flooding increase because you have an additional volume of liquid equal to the draw rate sitting on the tray. This percentage can be equivalent for all trays with draws, you cannot specify different percentages for different draws on different trays.

If you have vapour feed(s) attached to your column, HYSYS can size that particular tray to which it is attached either using the vapour feed to the tray section, or the vapour flow leaving it. You can specify which method HYSYS uses from the Use Tray Vapour to Size drop-down list.

The selection of method in the Use Tray Vapour to Size drop-down list affects all trays, regardless of whether they have a vapour feed or even any feed at all. The difference in method is its selection of the vapour that comes from the tray below or the vapour that leaves the tray as a basis for sizing calculations.

The table below contains the three selections from the Use Tray Vapour to Size drop-down list and their descriptions:

Selection	Calculation Method
Always Yes	HYSYS uses the vapour flow leaving the tray section to size the tray to which a vapour feed is attached for all calculations. The effect of the feed on the tray sizing is considered. This method generates results that closely represent reality. For example, If you have a vapour feed to your column, you can choose which tray the feed vapour is taken into account in the sizing/flooding calculations. If you have a vapour feed to tray 22 and you select “Always Yes”, HYSYS takes the vapour feed and assumes that it is in equilibrium with the vapour underneath tray 22. The vapour feed plus the vapour from tray 23 are used in sizing tray 22.
Always No	HYSYS uses the vapour feed to the section to size the tray for all calculations. The effect of the feed is not considered for this method. For example, If you have a vapour feed to tray 22 and you select “Always No”, HYSYS assumes that the vapour feed is in equilibrium with the vapour leaving tray 22. The assumed vapour feed is used in the sizing calculations fro Tray 21.
Ask Each Time	Prior to calculating the Tray Sizing utility, HYSYS asks you to specify whether to use feed to or vapour flow from the tray as a basis. A message property view for each vapour feed to your column appears prior to the calculations, as shown the in the figure below: 

In addition to the start and end stages, the following information is available for each section:

- Type of tower internals
- Calculation mode
- Sizing calculation information

Using Auto Section

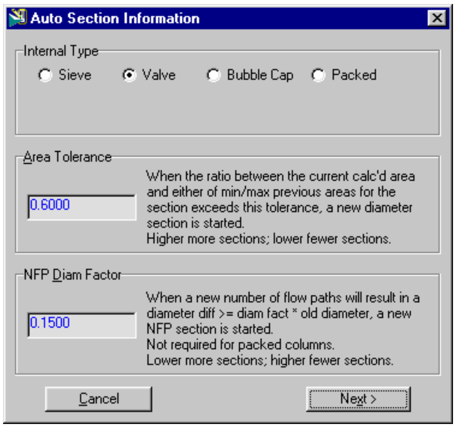
Refer to **Section 14.19.4 - Auto Section** for more information about the Auto Section feature.

The Auto Section feature in HYSYS provides a good starting point for the tray section analysis. The feature creates tower sections of constant diameter based on the parameters you specified.

The following steps shows you how to attach the main tray section to the utility and use the Auto Section functionality to divide the column into sections:

1. Select the column trays you want to size.
2. Click the **Auto Section** button. The Auto Section Information property view appears.

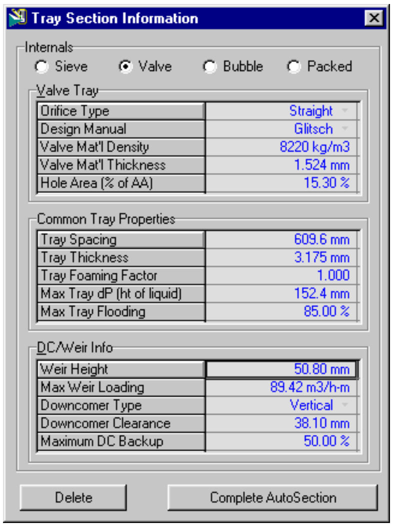
Figure 14.125



3. In the Internal Type group, select the type of tray the column contains using the radio buttons.
4. In the Area Tolerance and NFP Diameter Factor group, HYSYS provides default values for area tolerance and NFP diameter factor. You can also enter the value you want in the field provided.

5. Click the **Next** button. The Auto Section Information property view closes, and the Tray Section Information property view appears.

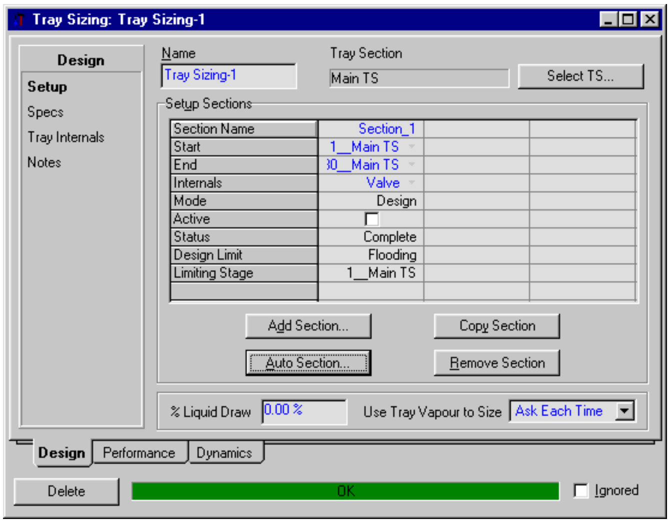
Figure 14.126



6. You can specify more details about the tray type in the group below the radio buttons. The group's name depends on which tray type your select.
7. In the Common Tray Properties group, you can specify the values for tray spacing, tray thickness, tray foaming factor, maximum tray dP, and maximum tray flooding.
8. In the DC/Weir Info group, you can specify the information for weir height, maximum weir loading, downcomer type, downcomer clearance, and maximum DC backup.
9. Click the **Complete AutoSection** button, when you are done editing the tray section information. HYSYS proceeds with the Auto Section calculations.

10. The Tray Section Information property view automatically closes, and you return to the **Setup** page of the Tray Sizing utility property view.

Figure 14.127



Adding Tray Sections

When using the Add Section button, HYSYS adds a new tray section covering the entire span of the column as the default size. This can be changed to a shorter span, if desired, by changing the start and end stages. A preliminary design calculation is automatically performed using all HYSYS default sizing parameters.

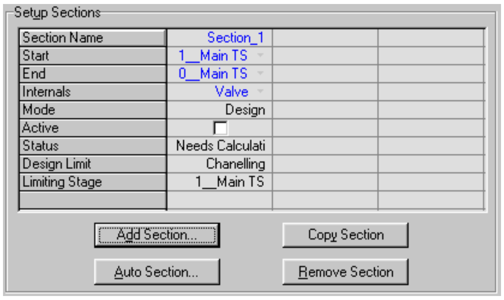
You can manually add the tray sections you want by performing the following steps:

1. Select the column trays you want to size.
2. Click the **Add Section** button.

   You can add more than one section by clicking the Add Section button again. Depending on the type of column you have, HYSYS displays warning property views that recommend what type of tray you should use for the column.

3. A section appears in the Setup Sections group.

Figure 14.128



4. The information displayed in the Setup Sections group are default values HYSYS provides. You can change the information to suit your scenario.

Copying a Tray Section

To copy a section perform the following steps:

1. Select the section you want to copy from the Setup Sections group.
2. Click the **Copy Section** button.
3. A copy of the selected section appears in the table.

Removing a Tray Section

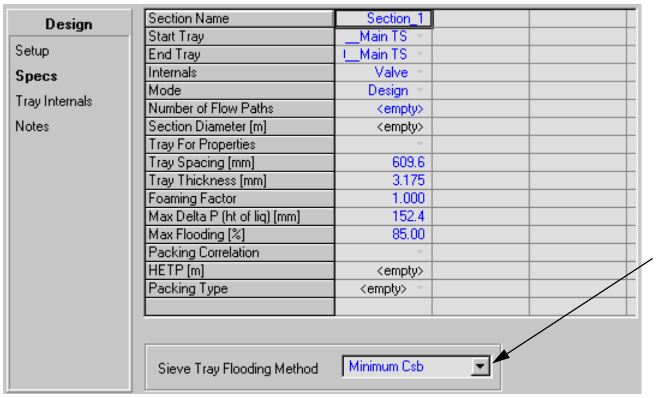
To remove a section perform the following steps:

1. Select the section you want to remove from the Setup Sections group.
2. Click the **Remove Section** button.
3. HYSYS removes the selected section from the Setup Sections group.

Specs Page

The Specs page allows you to specify the column internals for each section.The options are arranged in a tabular format.

Figure 14.129



You can select the sieve tray flooding method using this drop-down list. Refer to the section on the **Sieve Tray Flooding Method** for more information.

The following tables outline the available design and tray configuration parameters for the sizing utility.

Design Parameters

Parameter	Trayed	Packed
Design Correlation		4
Foaming Factor	4	4
Flooding	4	4
Pressure Drop	4	4
Downcomer Backup	4	
Weir Loading	4	

Tray Configuration Parameters

Parameter	Valve	Sieve	Bubble
Number of Flow Paths	4	4	4
Tray Spacing	4	4	4
Tray Thickness	4	4	4
Weir Height	4	4	4
Downcomer Type	4	4	4
Downcomer Clearance	4	4	4
Design Manual	4		
Hole Area	4		4
Hole Diameter		4	
Hole Spacing		4	
Hole Pitch		4	
Valve Density	4		
Valve Thickness	4		
Orifice Type	4		
Bubble Cap Slot Height			4

The Section Name, Start and End Trays, Internals, and Mode from the table on this page are common with the Setup page. However, it is only on the Specs page that you are able to change the calculation mode for the section. The following sections describes some of the options in the table.

Number of Flow Paths

This value represents the number of independent flow paths per tray. Usually a smaller tower diameter can be obtained by using multi-pass trays. However, with more flow paths, there is a reduction in the number of valves or sieve holes that can be placed on the tray. This can result in an increase in the pressure drop, an increase in downcomer backup, and a loss in tray efficiency. The following are general guidelines relating the number of flow paths and the tower diameter:

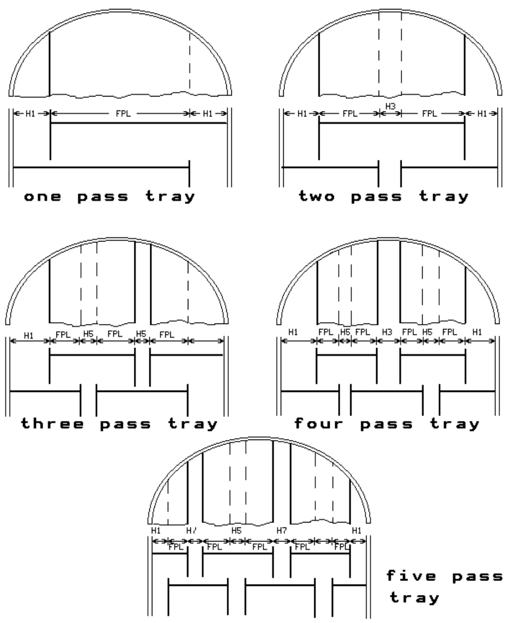
Number of Passes	Min. Diameter (ft)	Pref. Diameter (ft)
2	5	6
3	8	9

Number of Passes	Min. Diameter (ft)	Pref. Diameter (ft)
4	10	12
5	13	15

If the number of flow paths is not specified, HYSYS starts at one pass and increases the number of passes until the minimum diameter for that number of flow paths is reached. If a smaller number of flow paths is required, a new value can be entered that overrides the calculated NFP. A new solution is calculated as soon as the new NFP is entered.

The figure below summarizes the basic physical layouts of the flow paths available.

Figure 14.130



Section Diameter

Displays the diameter of the tray section based on the number of flow paths specified.

Tray for Properties

This cell is available only in Rating mode. You can specify which tray is used to calculate properties for the column.

Tray Spacing

The tray spacing is the vertical distance between trays. Some general guidelines for tray spacing follow:

Expected Tower Diameter (ft)	Suggested tray Spacing (in)
---	12 (minimum)
Up to 4	18 - 20
4 - 10	24
10 - 12	30
12 -24	36

The default value is 24 inches.

Valve and Tray Material Thickness

Since material thickness is often described in terms of gauge, the following table is provided for quick conversions between gauge and inches:

Gauge	Thickness (in)
20	0.037
18	0.050
16	0.060
14	0.074
12	0.104
10	0.134

The default tray thickness is 0.125 inches.

Foaming Factor

Foaming Factor is a measure of the foaming tendency of the system. In general, a lower foaming factor results in a lower overall tray efficiency and requirements for a larger tower diameter.

General Foaming Classification	Foaming Factor
Non Foaming Systems	1.00
Low Foaming Tendencies	0.90
Moderate Foaming Tendencies	0.75
High Foaming Tendencies	0.6

Foaming factors typically seen in some common systems include the following:

Absorbers	Foaming Factor
Ambient Oil (T > 0°F)	0.85
Low Temp Oil (T < 0°F)	0.95
DGA/DEA/MEA Contactor	0.75
Glycol Contactor	0.65
Sulfinol Contactor	1.0

Crude/Vacuum Tower	Foaming Factor
Crude or Vacuum Fractionation	1.00

Fractionators	Foaming Factor
Hydrocarbon	1.00
Low MW Alcohols	1.00
Rich Oil DeC1 or DeC2 (top)	0.85
Rich Oil DeC1 or DeC2 (Btm)	1.0
Refrigerated DeC1 or DeC2 (top)	0.80
Refrigerated DeC1 or DeC2 (btm)	1.00
General Hydrocarbon Distillation	1.00
MEA/DEA Still	0.85
Glycol/DGA Still	0.80
Sulfinol Still	1.00
H_2S Stripper	0.90
Sour Water Stripper	0.50 - 0.70
O_2 Stripper	1.00

Maximum Pressure Drop

The maximum allowable pressure drop per tray can be entered as a height of liquid. If it is not specified, a default maximum of 4 inches of liquid is used. For packed sections, the specification is on a pressure drop per height of packing basis. The default specification is 0.5 inches of water per foot of packing.

Maximum Flooding

The column is sized such that for the given vapour and liquid traffic, the tower flooding not exceed this specification on any stage. The maximum recommended value is 85% for normal service and 77% for vacuum or low pressure drop applications. These values yield approximately 10% entrainment. For diameters under 36 inches a reduced flooding specification of 65 - 75% should be used. A lower value can be specified to allow for contingencies, such as increased capacity. If not specified, a maximum flood factor of 82% is used for flat orifice trays and 77% for venturi orifice trays.

Packing Correlation

The Robbins correlation, which is the default selection, is noted to be better at predicting pressure drop and liquid holdup, particularly with newer packing materials. It is valid only at loading factors < 20000 (liquid loading < 9200 lb/hr.ft^2). The SLE (Sherwood-Leva-Eckert) correlation should be selected for towers operating above this range.

The last three rows are only available for packed towers.

Packed tower information can be specified in the last three parameters. Default values are provided for all packing parameters with the exception of the packing type and the packed section diameter.

HETP

The height factor HETP relates packed towers and tray towers. The value refers to the height of packing that is equivalent to a theoretical plate. For design purposes, the most accurate HETP factors are those published by packing manufacturers.

Packing Material

The packing type can be accessed on the Tray Internals page of the utility. A list of the available packing types is shown in the following table.

Packing Type	Material	Packing Type	Material
Ballast Rings	M,P	Jaeger_VSP_SS	M
Ballast Plus Rings	M	Koch-Sulzer(BX) Structured	S
Ballast Saddles	P	Lessing Experimental	M
Berl Saddles	C	Levapacking	P
Cascade MiniRing	M,P,C	Maspak	P
Chempak	M	Montz A-2 Structured	S
Flexipac Mellapac	S	Neo-Kloss Structured	S
Flexirings	M	Norton Intalox Metal Tower Packing	M
Gempak	S	Nutter Rings	M
Glitsch Grid	S	Pall Rings	M,P
Goodloe	S	Protruded	M
Wire Coil Packing	M	Raschig Rings 1/32 in wall	CSteel
Hy-Pak Rings	M	Raschig Rings 1/16 in wall	CSteel
Hyperfil	S	Raschig Rings	C, Carbon
Intalox Saddles	C	Super Intalox Saddles	P
Jaeger MaxPack SS	M	Tellerettes	P
Jaeger Tripacks	P	Cross-Partition Rings	C

The materials used for the different packings (unless otherwise noted) are: Metal(M), Plastic(P), Ceramic(C), and Metal Structured(S).

Sieve Tray Flooding Method

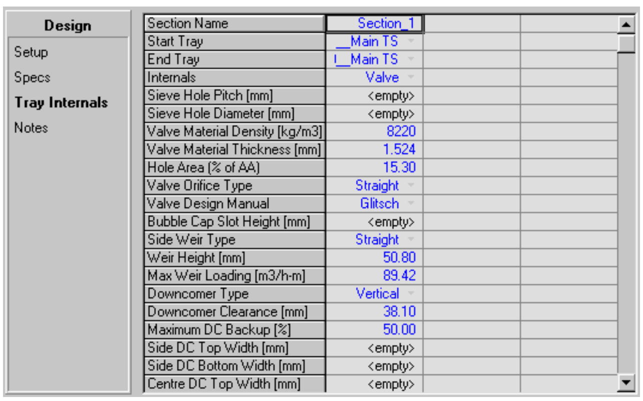
The method used to model sieve tray flooding can be specified using the drop-down list at the bottom of the Specs page. The options are:

- Minimum Csb
- Original Csb
- Fair’s Modified Csb

Tray Internals Page

If the sizing section is specified as having trayed internals (Sieve, Valve or Bubble Cap**)**, then the internals can be further specified on the Tray Internals page. There are certain column parameters that are common to all trayed columns. You can specify these parameters, or leave them at their default values.

Figure 14.131



Sieve Tray Parameters

The input available for the configuration of the sieve tray is similar to the valve tray with the exception of the tray input data.

Hole Pitch

Hole pitch refers to the distance between the centers of two adjacent holes. HYSYS requires the hole pitch to be within 1.5 to 5 times the hole diameter. The default hole pitch is 0.5 inches.

Hole Diameter

The default value for the hole diameter is 0.187 inches.

Valve Tray Parameters

The valve tray is the default tray type for trayed columns in design mode. Defined here are the design parameters specific to valve trays:

Valve Material Density

A table of typical materials used for valves and their associated densities follows:

Valve Material	Density (lb/ft^3)
Carbon Steel	480
Stainless Steel	510
Nickel	553
Monel	550
Titanium	283
Hastelloy	560

Hole Area (% of Active Area)

The hole area is the percentage of the active area that is occupied by the valve holes. The default is 15.3%, which corresponds to 12 valves, each having a diameter of 1 17/32 inches, per square foot.

The Hole Area can also be specified for Bubble Cap trays.

For bubble cap trays, the default hole area is also 15.3%, which corresponds to 12 bubble caps, each having a diameter of 1 17/32 inches, per square foot.

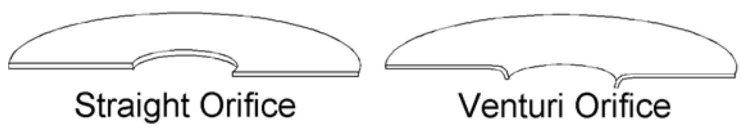
Valve Orifice Type

The Valve Orifice is the shape of the hole that is punched in the plate for the valve. As shown in the figure below, there are two types of orifices:

- Venturi
- Straight

The straight orifice is used for normal service, while the Venturi orifice is used for low pressure drop applications.

Figure 14.132



Design Manual for Flooding Calculations

Results are presented for flooding calculations from three industry standard design manuals (Glitsch, Koch or Nutter). Any one of the three methods can be selected as the basis for comparison with the maximum allowable % of flood design specification.

The default design manual is Glitsch.

Bubble Cap Trays

Bubble Cap Slot Height

The default slot height is 1.0 inch. This value represents the height of the slots around the base of the bubble caps, through which the gas and liquid are allowed to flow.

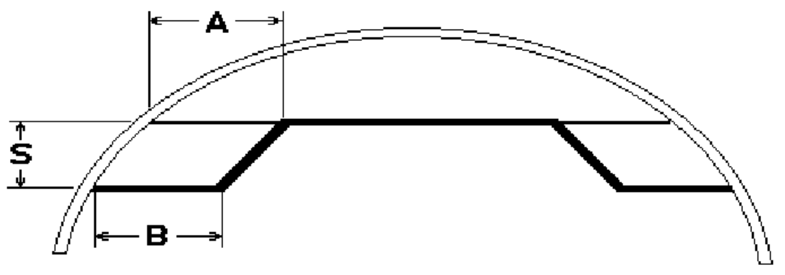
Common Tray Parameters

Side Weir Type

This parameter is used to specify the side weir type only. There are two types of weirs available: straight and relief. A relief weir lengthens the side weir without increasing the downcomer area.

The relief weir sweeps back, then across the tray, enclosing some active area, as shown in the figure below. A straight weir follows the edge of the downcomer.

Figure 14.133



A relief weir is used for high liquid loads or where a low pressure drop is required, while straight weirs are used for normal service. HYSYS uses a straight weir as the default. However, if the weir loading is above the maximum, a relief weir can be included to alleviate the problem. If a relief weir is installed by HYSYS and a straight weir is desired, a straight weir can be re-specified and the tray rerun in *rating* mode.

Weir Height

The weir height is the distance from the tray to the top of the weir. A weir height of 2 inches is used in most applications. However, a smaller height can be used for low pressure drop or vacuum services. A larger weir height is used to obtain longer residence times (for example chemical reaction services).

The default value for weir height is 2 inches. In general, you can use the following:

Tray Spacing (in)	Weir Height (in)
12	1.5
12 - 24	2
>24	2.5

Maximum Allowable Weir Loading

The weir loading is a measure of the liquid loading on the weirs. Values of 60 - 120 USGPM/ft are typical. Weir loading may be reduced by increasing the number of flow paths or installing a relief weir. A weir loading as high as 240 USGPM/ft can sometimes be tolerated. If the weir loading is not specified, a default value of 96 USGPM/ft is used.

Downcomer Type

There are two types of downcomers available:

- vertical
- sloped

A sloped downcomer has a narrower width at the bottom. This allows more active area and more valves per tray, and also results in a lower pressure drop. Due to cost considerations a vertical downcomer is used for normal service and is the default in HYSYS.

Downcomer Clearance

The downcomer clearance is the distance between the bottom of the downcomer and the tray. The area available for liquid flow under the downcomer is dependent upon this height.

A minimum seal of 0.5 inches is normally recommended. For high liquid velocities and the resulting high pressure drop, this can be reduced. If the downcomer clearance is not specified, a height of 0.5 inches less than the weir height is used.

Since the weir height default is 2 inches, this translates to a downcomer clearance default of 1.5 inches.

Maximum Allowable Downcomer Backup

The allowable downcomer backup is measured as the percentage of the tray spacing that the liquid level in the downcomer is allowed to reach. This value represents the average for all the downcomers on the tray. If not specified, a value of 40% is used for services with a vapour density greater than 3 lb/ft^3, 50% for normal densities, and 60% for densities less than 1 lb/ft^3.

Refer to **Column Rating** for more information.

The remaining fields on this page are available only when in Rating mode.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page/Tab**.

The Notes page provides a text editor, where you can record any comments or information regarding the utility, or to your simulation case in general.

14.19.2 Performance Tab

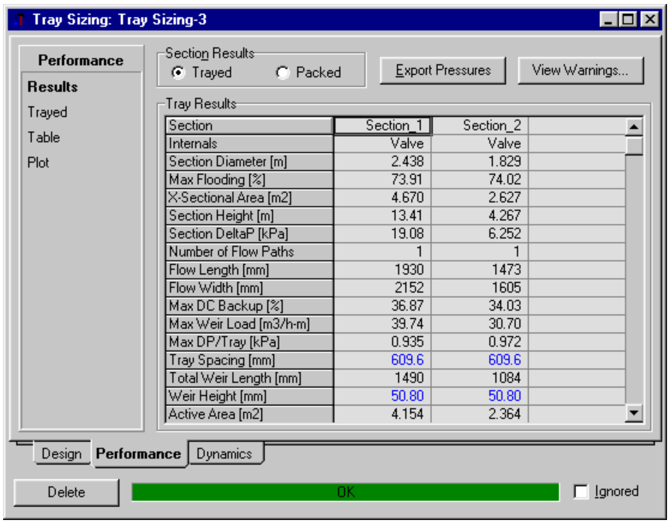
The Performance tab contains the following pages:

- **Results**. Overall comparative section results and detailed tray sizing information.
- **Trayed**. Pressure Drop, Downcomer, and Flooding results across the column.
- **Table**. Tray section physical property profiles in the tabular form.
- **Plots**. Tray section physical property profiles in the graphical form.

Results Page

The Results page contains a more detailed description of the tray section outputs.

Figure 14.134



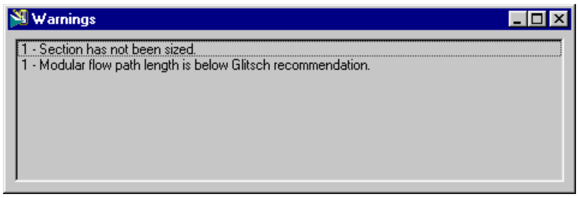
The Section Results group has two radio buttons:

- **Trayed**. Selecting this radio button tells HYSYS to calculate the tray size for a tray column/tower.
- **Packed**. Selecting this radio button tells HYSYS to calculate the tray size for a packed column/tower.

Clicking the Export Pressure button signals HYSYS to take the calculated pressure profile and export it to the active tray section, thus causing the column to reconverge to the pressure profile predicted by the tray sizing utility. A section in the utility must be made active, before this option can be used.

Click the **View Warnings** button to see any problems HYSYS detects in the tray section. The figure below shows two possible warning messages if the incorrect tray type is selected.

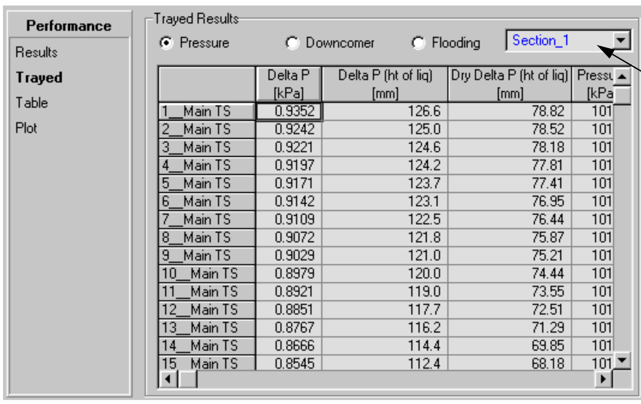
Figure 14.135



Trayed Page

On the Trayed page, HYSYS displays tray-by-tray information for the selected section.

Figure 14.136



	Delta P [kPa]	Delta P (ht of liq) [mm]	Dry Delta P (ht of liq) [mm]	Pressu [kPa
1__Main TS	0.9352	126.6	78.82	101
2__Main TS	0.9242	125.0	78.52	101
3__Main TS	0.9221	124.6	78.18	101
4__Main TS	0.9197	124.2	77.81	101
5__Main TS	0.9171	123.7	77.41	101
6__Main TS	0.9142	123.1	76.95	101
7__Main TS	0.9109	122.5	76.44	101
8__Main TS	0.9072	121.8	75.87	101
9__Main TS	0.9029	121.0	75.21	101
10__Main TS	0.8979	120.0	74.44	101
11__Main TS	0.8921	119.0	73.55	101
12__Main TS	0.8851	117.7	72.51	101
13__Main TS	0.8767	116.2	71.29	101
14__Main TS	0.8666	114.4	69.85	101
15__Main TS	0.8545	112.4	68.18	101

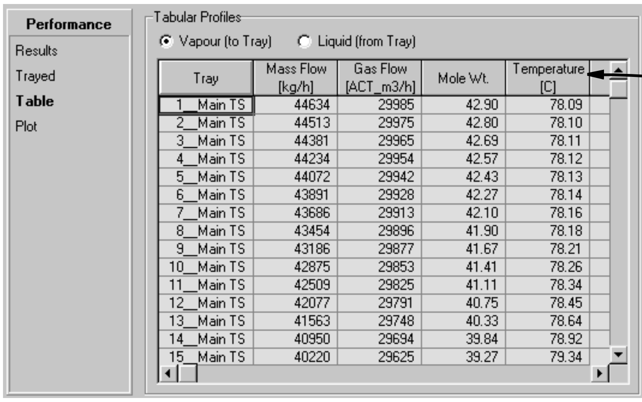
You can select different sections using the drop-down list.

By selecting the corresponding radio buttons, information on Pressure Drop, Downcomer, or Flooding can be displayed for each tray section.

Table Page

The Table page contains two radio buttons (Vapour to Tray and Liquid from tray), and a table that displays the values of six of column variables for each tray.

Figure 14.137



Tray	Mass Flow [kg/h]	Gas Flow [ACT_m3/h]	Mole Wt.	Temperature [C]
1__Main TS	44634	29985	42.90	78.09
2__Main TS	44513	29975	42.80	78.10
3__Main TS	44381	29965	42.69	78.11
4__Main TS	44234	29954	42.57	78.12
5__Main TS	44072	29942	42.43	78.13
6__Main TS	43891	29928	42.27	78.14
7__Main TS	43686	29913	42.10	78.16
8__Main TS	43454	29896	41.90	78.18
9__Main TS	43186	29877	41.67	78.21
10__Main TS	42875	29853	41.41	78.26
11__Main TS	42509	29825	41.11	78.34
12__Main TS	42077	29791	40.75	78.45
13__Main TS	41563	29748	40.33	78.64
14__Main TS	40950	29694	39.84	78.92
15__Main TS	40220	29625	39.27	79.34

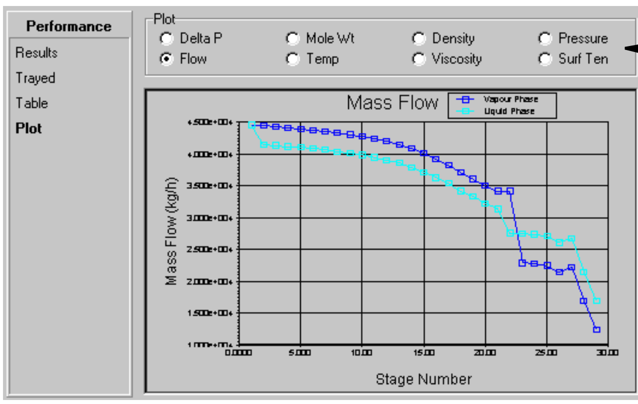
The column variables are:

- Mass Flow
- Gas/Liquid Flow
- Molecular Weight
- Temperature
- Density
- Viscosity

Plot Page

The Plot page displays plots for a number of column variables.

Figure 14.138

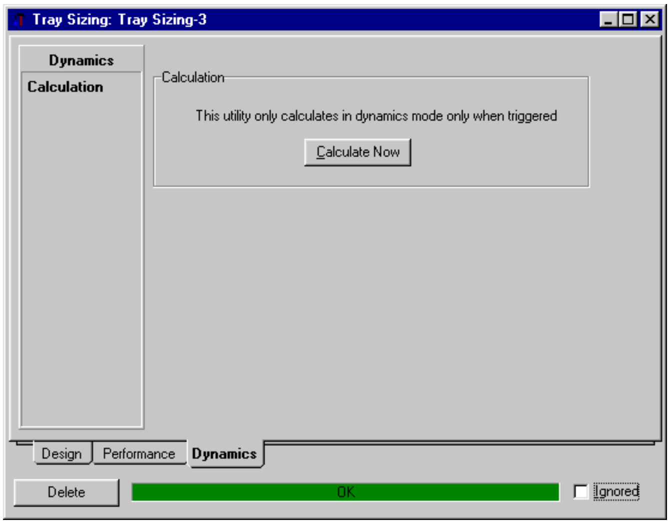


Select the column variable you want to plot using the radio buttons in the Plot group.

14.19.3 Dynamics Tab

The Dynamics tab contains the Calculation page.

Figure 14.139



The Calculation page contains the Calculate Now button. Clicking the Calculate Now button causes the Tray Sizing utility to calculate when you are in Dynamic mode. We do not do any calculations by default in dynamics, so if you want to see the results while in Dynamic mode you have to click this button.

14.19.4 Auto Section

The Auto Section function is an optional method in design mode. When using Auto Section, HYSYS automatically calculates the sections for a column. They are then transferred onto the main utility Setup tab where you can edit, copy, or delete sections. Auto Section provides you with an excellent starting point in the design of a tower by performing a summary sizing of the tray section and splitting it into sections of constant diameter, as appropriate.

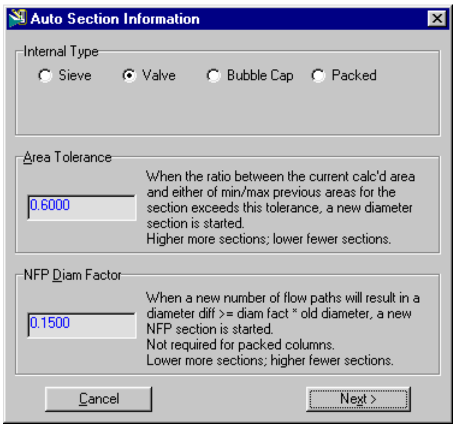
The Auto Section routine is not available until a tray section has been selected on the Setup tab of the tray sizing utility.

The sections that HYSYS determines during its analysis are automatically available for you to edit, rename, or delete.

HYSYS allows you to specify the internal type, as well as values for the criteria that are used to start a new section. The two criteria that the user can specify to establish tower sections are:

- Area Tolerance
- NFP Diameter Factor

Figure 14.140



Area Tolerance

The Area Tolerance defines the magnitude of change in the calculated area that causes the start of a new section. HYSYS first performs a design for stage i, using the current parameters for the chosen internals, and the NFP for the current section (valve, sieve, and bubble trays only) to determine the required area. This area is compared to the minimum and maximum areas for the current section, which HYSYS retains.

If the magnitude of difference for either comparison exceeds the Area Tolerance, a new section is started beginning at stage i.

The previous section can be assigned the maximum diameter for that section. If the calculated area for stage i is outside the range defined by the minimum and maximum for the section but does not exceed the tolerance, the calculated area for stage i replaces the appropriate stored value.

NFP Factor

When the comparison of areas is complete, HYSYS recalculates the required area of each stage using a different Number of Flow Paths. This area is compared with the previously calculated area for each stage. If the magnitude of the change is greater than the NFP Diam Factor, a new section is started.

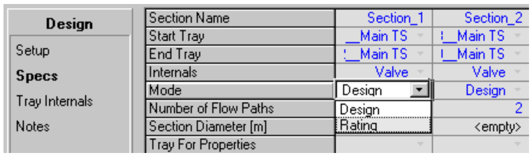
The entire column is stepped off in the above manner according to the Area and NFP guidelines and sections of constant diameter, number of flow paths, active area, and downcomer area are defined for the tower. When this initial sizing is complete, HYSYS re-rates each tray based on a diameter calculated from the maximum downcomer and active area required for trays in that section. This value is available on the Results page of the Performance tab of the utility property view. The limiting factor(s) for each section appear on the Setup tab.

Using Auto Section, you are required to specify the tray internal type, either Sieve, Valve, Bubble Cap or Packed. In addition, you have the option of specifying the parameters for the chosen configuration. If no parameters are specified, HYSYS uses values from its default set.

Column Rating

Column sections can be rated using the Rating calculation mode on the Specs page. From the Mode drop-down list, select Rating, as shown in the figure below.

Figure 14.141



If you modify the Main Flowsheet or Column subflowsheet, the tray sizing utility redesigns and rerates all of the sections based on the current configuration using the new stage by stage traffic, physical, and transport property information from the Column subflowsheet.

Trayed Section Rating

For trayed sections, the minimum required information for HYSYS to calculate the section performance includes the number of flow paths and the column diameter.

If desired, you can specify the following downcomer widths on the Tray Internals page:

- Side top and optional bottom
- Centre top and optional bottom
- Off-side top and optional bottom
- Off-centre top and optional bottom

The optional bottom width allows for the specification of sloped downcomers. Downcomer widths that are not specified can be calculated at optimal design values for the given number of flow paths.

The remaining tray configuration parameters can be specified as discussed in the **Specs Page** section. Once rating parameters have been set, HYSYS completes the rating calculations.

Packed Section Rating

For packed sections, the required information for HYSYS to calculate the section performance includes both the Section Diameter and the Tray for Properties on the Tray Internals page.

The remaining tray configuration parameters can be specified as discussed in the **Specs Page** section. Once rating parameters have been set, HYSYS completes the rating calculations. If HYSYS is unable to complete the calculations, this will be indicated on the property view's status bar. To view the warnings generated, click on the Results page of the Performance tab and click the **View Warnings** button.

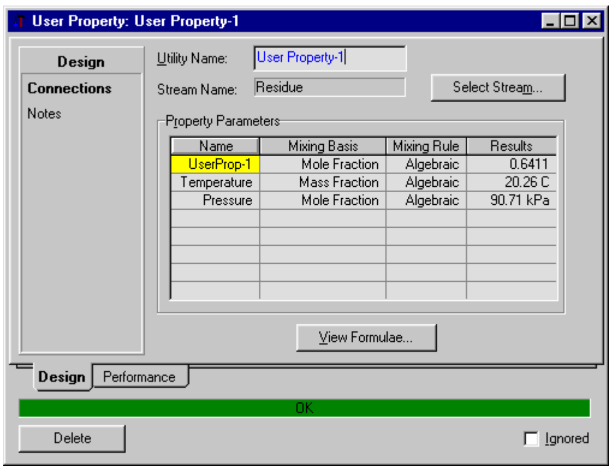
14.20 User Properties

Refer to **Chapter 7 - User Properties** of the **HYSYS Simulation Basis** guide for detailed information concerning User Properties.

The User Property utility allows you to view User Properties (that you have defined) based on the composition of a stream. You can only add User Properties in the Basis environment.

Possible uses of the User Property include as a specification in a distillation column or as a target variable in an Adjust operation.

Figure 14.142



14.20.1 Design Tab

To add the User Properties utility, refer to the section on **Adding a Utility**.

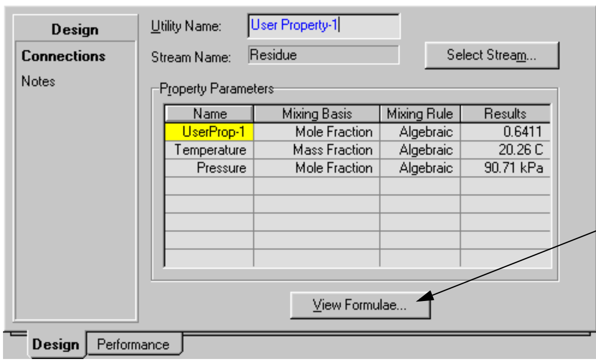
The Design tab contains the following pages:

- Connections
- Notes

Connections Page

You can specify the stream you want to attach to the utility, and add the properties you want on the Connections page.

Figure 14.143



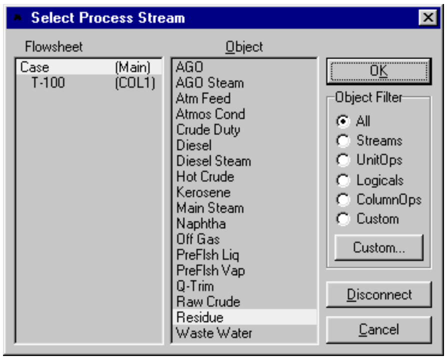
Click the **View Formulae** button to view the selected Mixing Rule formula.

You can also change the name of the utility on this page, if desired.

Adding a Stream to User Properties

1. On the **Design** tab, click on the **Connections** page.
2. Click the **Select Stream** button. The Select Process Stream property view appears.

Figure 14.144



3. Select a stream from the Object list, and click the **OK** button. You return to the **Connections** page.

You can disconnect a stream by clicking the Disconnect button in the Select Process Stream property view.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page/Tab**.

The Notes page provides a text editor, where you can record any comments or information regarding the utility, or to your simulation case in general.

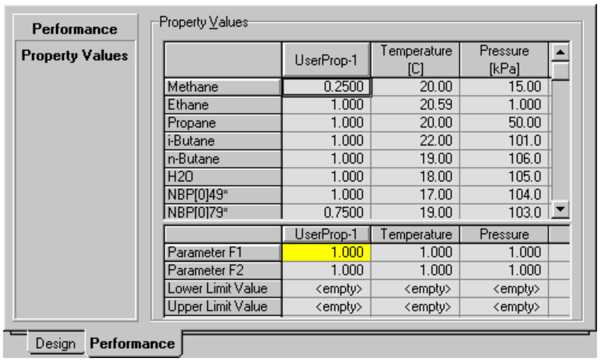
14.20.2 Performance Tab

The Performance tab contains the Property Values page.

Property Values

The Property Values page contains two tables.

Figure 14.145



The top table contains a list of all the components in the stream. The bottom table contains the parameter values for the equations to use on the Connections page.

It is recommended that you leave the mixing rule equation parameters at their default values.

Refer to **Section 7.3.1 - Data Tab** in the **HYSYS Simulation Basis** guide, for information about the parameter values available for you to manipulate.

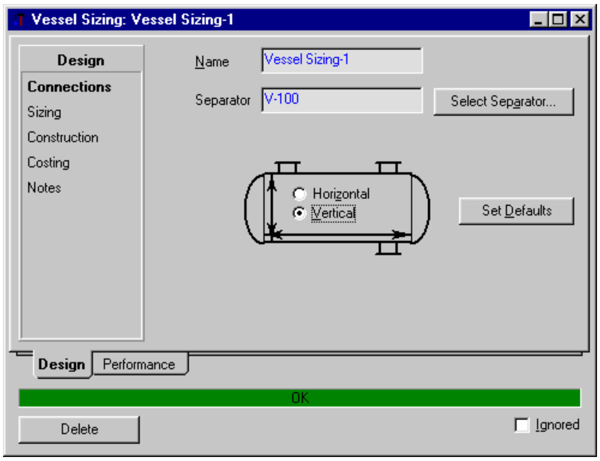
You can only change the parameter values of the property in the Basis Environment.

14.21 Vessel Sizing

To add the Vessel Sizing utility, refer to the section on **Adding a Utility**.

The Vessel Sizing utility allows you to size and cost installed separator, tank, and reactor unit operations. You can select a vertical or horizontal orientation for the separator. To obtain a more effective analysis for your vessel, changes can be made to the default parameters provided by HYSYS.

Figure 14.146



For a comprehensive costing and sizing software package for your entire case, Economix is available. Contact your nearest AspenTech office or agent for details.

14.21.1 Design Tab

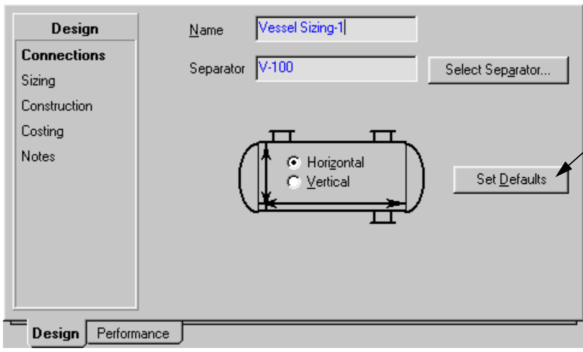
The Design tab allows you to select the vessel that is to be sized, the dimensions of the vessel, the vessel material, and the cost of the vessel. The tab consists of the following pages:

- Connections
- Sizing
- Construction
- Costing
- Notes

Connections Page

On the Connections page, you select the vessel you want to size and the vessel's orientation. You can also change the name of the utility on this page.

Figure 14.147

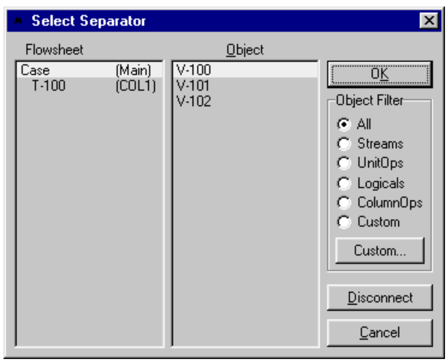


Clicking the Set Defaults button returns **all** of the original data provided by HYSYS.

To select a vessel:

1. On the **Design** tab, click on the **Connections** page.
2. Click the **Select Separator** button. The Select Separator property view appears.
3. Select the vessel you want to size from the Object list, and click the **OK** button.

Figure 14.148

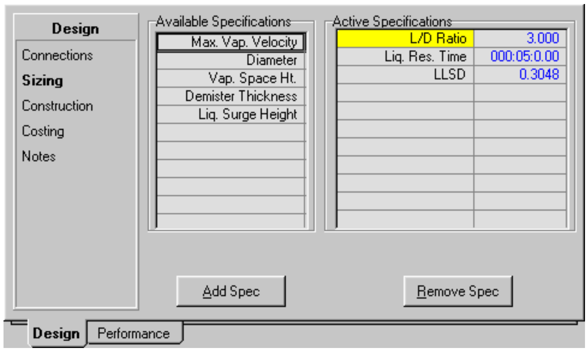


4. You automatically return to the **Connections** page. Now select the orientation of the vessel using the **Vertical** or **Horizontal** radio button.

Sizing Page

The Sizing page allows you to set the specification variables that are used to size the vessel.

Figure 14.149



To select the specification variable:

1. Select the specification variable from the Available Specification group.
2. Click the **Add Spec** button. The specification variable is moved into the Active Specification group.
3. HYSYS provides default values for the specification, but you can change the values. Enter the value you want for the specification variable in the cell provided.

   The following is the list of available specifications that are specific to the orientation of the separator:

   - Max. Vapour Velocity
   - Diameter

If the diameter value is not specified, HYSYS automatically changes this value when the vessel orientation is switched.

   - L/D Ratio
   - Vapour Space Height

- Demister Thickness
- Liquid Residence Time
- Liquid Surge Height
- Total Length - Height
- Nozzle to Demister
- Demister to Top
- LLSD (Low Level Shut Down)
- Total Separator Height

To remove the specification variable:

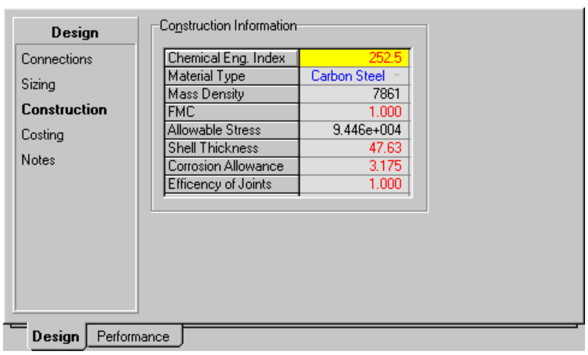
1. Select the specification variable you want to remove from the Active Specification group.
2. Click the **Remove Spec** button. The specification variable is moved back into the Available Specification group.

Construction Page

On the Construction page, you can specify any of the following information for the vessel:

- Chemical Engineering Index
- Material Type: Carbon Steel, SS 304, SS 316, Aluminium
- Mass Density
- FMC (material of fabrication factor)
- Allowable Stress
- Shell Thickness
- Corrosion Allowance

Figure 14.150



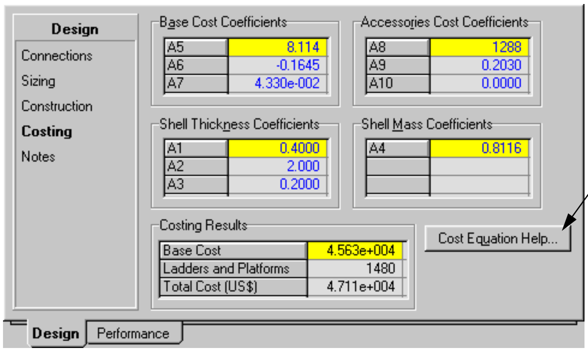
HYSYS recalculates after each change is made on the Construction page.

Blue text is entered by the user, and red text is entered by HYSYS. You can modify the blue and red text.

Costing Page

You can modify the factors used in the sizing and cost equations on the Costing page. These factors deal with the Base Cost, Shell Thickness, Accessories Cost, and Shell Mass.

Figure 14.151



View the various equations by clicking the Cost Equation Help button.

You can also view the results of the cost analysis on this page in the Costing Results group. The Base Cost, Ladders and Platform Cost, and Total Cost are listed in $US.

Notes Page

For more information, refer to **Section 1.3.5 - Notes Page/Tab**.

The Notes page provides a text editor, where you can record any comments or information regarding the utility, or to your simulation case in general.

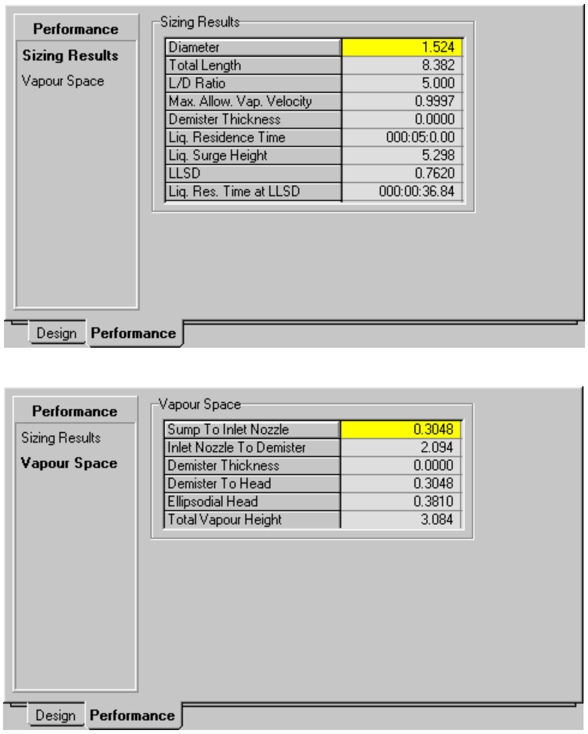
14.21.2 Performance Tab

The Performance tab contains the following pages:

- Sizing Results
- Vapour Space

A summary of the sizing results and vapour space are provided on the each page respectively.

Figure 14.152



14.22 References

[1] Marsland, R.H., “A User Guide on Process Integration for the Efficient Use of Energy”, Insitution of Chemical Engineers, England, 1982.

[2] Ng, H.J., Robinson, D.B., Ind Eng Chem Fund., 15, 293 (1976)

[3] Ng, H.J., Robinson, D.B., "The Measurement and Prediction of Hydrate Formation in Liquid Hydrocarbon-Water Systems", Ind. Eng. Chem. Fund.,15, 293 (1976)

[4] Ng, H.J., Robinson, D.B., AIChE J., 23, 477 (1977)

[5] Ng, H.J., Robinson, D.B., Ind Eng Chem Fundam, 19, 33 (1980).

[6] Ng, H.J., Robinson, D.B., "A Method for Predicting the Equilibrium Gas Phase Water Content in Gas-Hydrate Equilibrium", Ind. Eng. Chem. Fund.,19, 33 (1980)

[7] Overa, Sverre O., & Stange, Ellen, & Salater, “Per, Determination of Temperatures and Flow Rates During Depressurization and Fire”, presented at the 72 Annual GPA Convention, March 15-17, 1993, San Antonio, Texas.

[8] Parrish, W.R., Prausnitz, J.M., I.E.C. Proc Des Dev, 11, 26 (1972).

[9] Sloan, E.D., Khoury, F., Kobayashi, R., I.E.C. Fundam, 15, 318 (1976).

[10] Sloan,Jr.,E.D.,Clathate Hydrates of Natural Gases, Macel Dekker, Inc., New York, 1989

[11] van der Waals, J.H., Platteuw, J.C., Advan Chem Phys, 2, 1 (1959).

Index

D

E

F

G

H

I

K

L

Q

R

V

W

Z