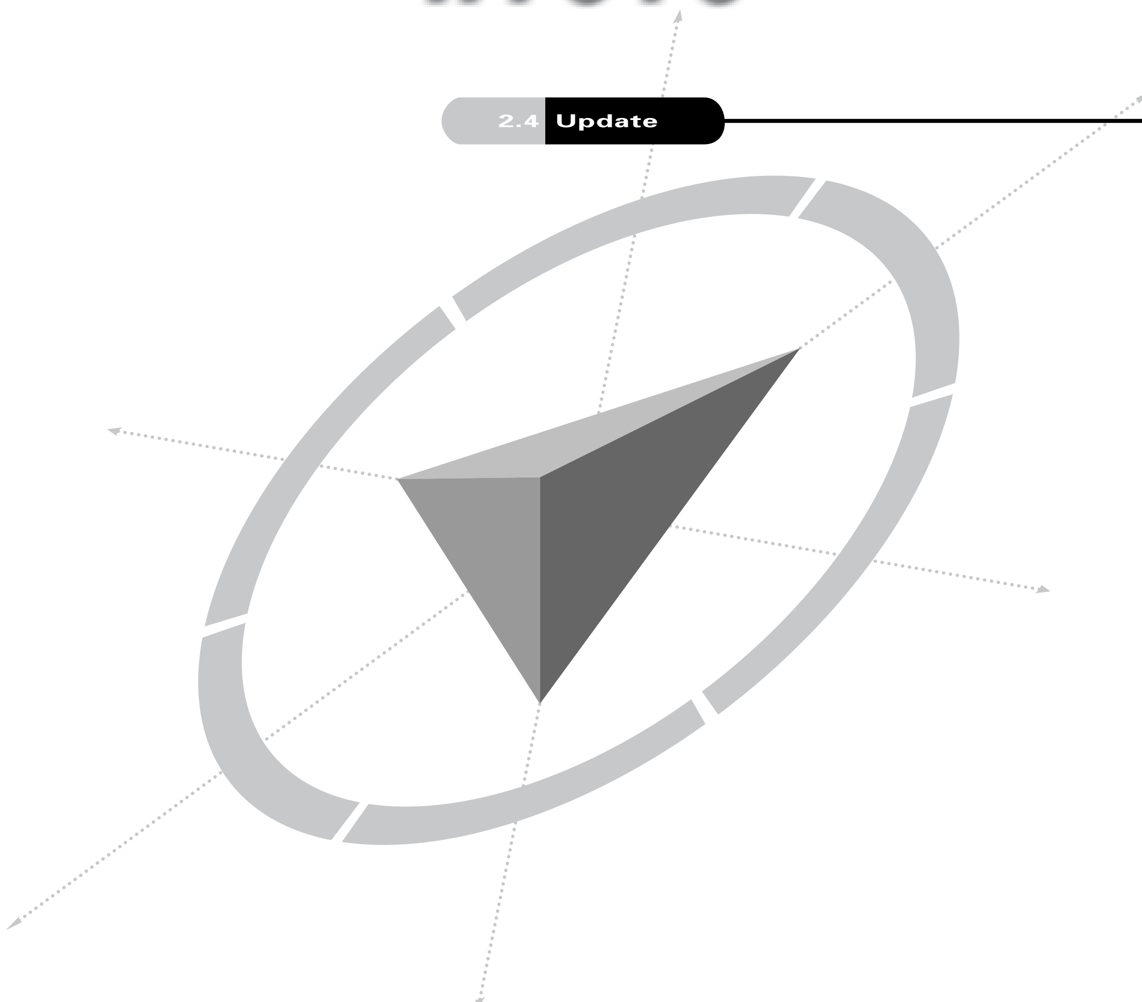
HYSYS

2.4 Update



AEA TECHNOLOGY
THE INNOVATION BUSINESS

Hyprotech is a member of the AEA Technology plc group of companies

Documentation Credits

Authors of the current release, listed in order of historical start on project:

Sarah-Jane Brenner, BASc; Conrad, Gierer, BASc; Chris Strashok, BSc; Lisa Hugo, BSc, BA; Muhammad Sachedina, BASc; Allan Chau, BSc; Adeel Jamil, BSc; Nana Nguyen, BSc; Yannick Sternon, BIng; Kevin Hanson, PEng; Chris Lowe, PEng.

Since software is always a work in progress, any version, while representing a milestone, is nevertheless but a point in a continuum. Those individuals whose contributions created the foundation upon which this work is built have not been forgotten. The current authors would like to thank the previous contributors.

A special thanks is also extended by the authors to everyone who contributed through countless hours of proof-reading and testing.

Contacting Hyprotech

Hyprotech can be conveniently accessed via the following:

Website:	www.hyprotech.com
Technical Support:	support@hyprotech.com
Information and Sales:	info@hyprotech.com

Detailed information on accessing Hyprotech Technical Support can be found in the **Technical Support** section in the preface to this manual.

Table of Contents

Welcome to HYSYS

We are pleased to present you with the latest version of ***HYSYS*** — the product that continually extends the bounds of process engineering software. With HYSYS you can create rigorous steady-state and dynamic models for plant design and trouble shooting. Through the completely interactive HYSYS interface, you have the ability to easily manipulate process variables and unit operation topology, as well as the ability to fully customize your simulation using its OLE extensibility capability.

Hyprotech Software Solutions

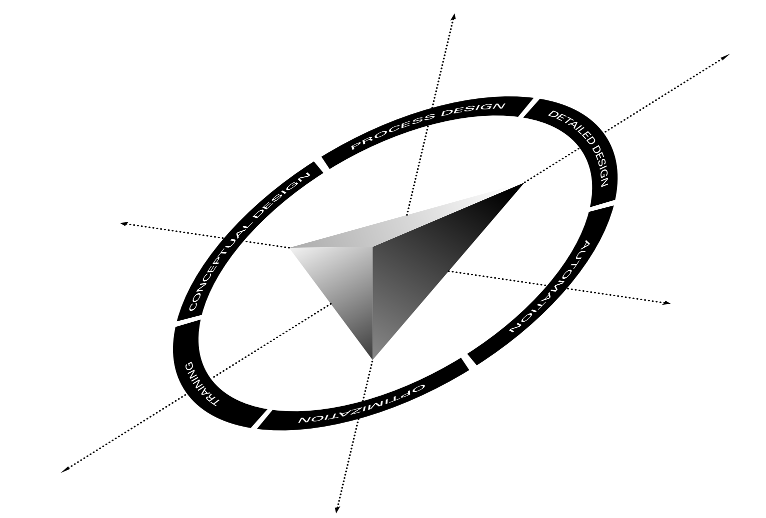
HYSYS has been developed with Hyprotech's overall vision of the ultimate process simulation solution in mind. The vision has led us to create a product that is:

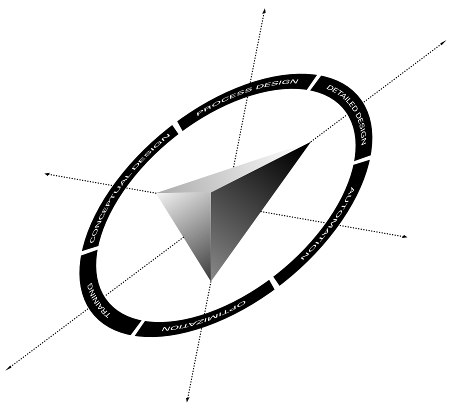
- Integrated
- Intuitive and interactive
- Open and extensible

Integrated Simulation Environment

In order to meet the ever-increasing demand of the process industries for rigorous, streamlined software solutions, Hyprotech developed the HYSYS Integrated Simulation Environment. The philosophy underlying our truly integrated simulation environment is conceptualized in the diagram below:

Figure 1





The central wedge represents the common parameters at the core of the various modelling tools:

- model topology
- interface
- thermodynamics

The outer ring represents the modelling application needs over the entire plant lifecycle. The arrows depict each Hyprotech product using the common core, allowing for universal data sharing amongst the tools, while providing a complete simulation solution.

As an engineer you undoubtedly have process modelling requirements that are not all handled within a single package. The typical solution is to generate results in one package, then transfer the necessary information into a second package where you can determine the additional information. At best, there is a mechanism for exchanging information through file transfer. At worst, you must enter the information manually, consuming valuable time and risking the introduction of data transfer errors. Often the knowledge you gain in the second application has an impact on the first model, so you must repeat the whole process a number of times in an iterative way.

In a truly integrated simulation environment all of the necessary applications work is performed within a common framework, eliminating the tedious trial-and-error process described previously. Such a system has a number of advantages:

- Information is shared, rather than transferred, among applications.
- All applications use common thermodynamic models.
- All applications use common flowsheet topology.
- You only need to learn one interface.
- You can switch between modelling applications at any time, gaining the most complete understanding of the process.

The plant lifecycle might begin with building a conceptual model to determine the basic equipment requirements for your process. Based on the conceptual design, you could build a steady-state model and perform an optimization to determine the most desirable operating conditions. Next, you could carry out some sizing and costing calculations for the required equipment, then do some dynamic modelling to determine appropriate control strategies. Once the design has become a reality, you might perform some online modelling using actual plant data for "what-if" studies, troubleshooting or even online optimization. If a change at any stage in the design process affects the common data, the new information is available immediately to all the other applications — no manual data transfer is ever required.

While this concept is easy to appreciate, delivering it in a useable manner is difficult. Developing this multi-application, information-sharing software environment is realistically only possible using Object Oriented Design methodologies, implemented with an Object Oriented Programming Language. Throughout the design and development process, we have adhered to these requirements in order to deliver a truly integrated simulation environment as the HYSYS family of products:

For information on any of these products, contact your local Hyprotech representative.

HYSYS Product	Description
HYSYS.Process	**Process Design** - ***HYSYS.Process*** provides the accuracy, speed and efficiency required for process design activities. The level of detail and the integrated utilities available in ***HYSYS.Process*** allows for skillful evaluation of design alternatives.
HYSYS.Plant	**Plant Design - *HYSYS.Plant*** provides an integrated steady-state and dynamic simulation capability, offers rigorous and high-fidelity results with a very fine level of equipment geometry and performance detail. ***HYSYS.Plant+*** provides additional detailed equipment configurations, such as actuator dynamics.
HYSYS.Refinery	**Refinery Modeling - *HYSYS.Refinery*** provides truly scalable refinery-wide modeling. Detailed models of reaction processes can be combined with detailed representations of separation and heat integration systems. Each hydrocarbon stream is capable of predicting a full range of refinery properties based on a Refinery Assay matrix.
HYSYS.OTS	**Operations Training System** - ***HYSYS.OTS*** provides real-time simulated training exercises that train operations personnel and help further develop their skills performing critical process operations. Increased process understanding and procedural familiarity for operations personnel can lead to an increase in plant safety and improvements in process performance.
HYSYS.RTO	**Real-Time Optimization - *HYSYS.RTO*** is a real-time optimization package that enables the optimization of plant efficiency and the management of production rate changes and upsets in order to handle process constraints and maximize operating profits.
HYSYS.Concept	**Conceptual Design Application - *HYSYS.Concept*** includes ***DISTIL*** which integrates the *distillation synthesis* and *residue curve map* technology of Mayflower with data regression and thermodynamic database access. ***HYSYS.Concept*** also includes ***HX-Net***, which provides the ability to use *pinch technology* in the design of heat exchanger networks. Conceptual design helps enhance process understanding and can assist in the development of new and economical process schemes.

Intuitive and Interactive Process Modelling

We believe that the role of process simulation is to improve your process understanding so that you can make the best process decisions. Our solution has been, and continues to be, interactive simulation. This solution has not only proven to make the most efficient use of your simulation time, but by building the model interactively – with immediate access to results – you gain the most complete understanding of your simulation.

HYSYS uses the power of Object Oriented Design, together with an Event-Driven Graphical Environment, to deliver a completely interactive simulation environment where:

- calculations begin automatically whenever you supply new information, and
- access to the information you need is in no way restricted.

At any time, even as calculations are proceeding, you can access information from any location in HYSYS. As new information becomes available, each location is always instantly updated with the most current information, whether specified by you or calculated by HYSYS.

Open and Extensible HYSYS Architecture

HYSYS is the only commercially available simulation platform designed for complete User Customization.

The Integrated Simulation Environment and our fully Object Oriented software design has paved the way for HYSYS to be fully OLE compliant, allowing for complete user customization. Through a completely transparent interface, OLE Extensibility lets you:

- develop custom steady-state and dynamic unit operations
- specify proprietary reaction kinetic expressions
- create specialized property packages.

With seamless integration, new modules appear and perform like standard operations, reaction expressions or property packages within HYSYS. The Automation features within HYSYS expose many of the internal Objects to other OLE compliant software like Microsoft Excel, Microsoft Visual Basic and Visio Corporation's Visio. This functionality enables you to use HYSYS applications as calculation engines for your own custom applications.

By using industry standard OLE Automation and Extension the custom simulation functionality is portable across Hyprotech software updates. The open architecture allows you to extend your simulation functionality in response to your changing needs.

Use of the Manuals

HYSYS Electronic Documentation

The HYSYS Documentation Suite includes all available documentation for the HYSYS family of products.

All HYSYS documentation is available in electronic format as part of the HYSYS Documentation Suite. The HYSYS Documentation CD ROM is included with your package and may be found in the Get Started box. The content of each manual is described in the following table:

Manual	Description
Get Started	Contains the information needed to install HYSYS, plus a Quick Start example to get you up and running, ensure that HYSYS was installed correctly and is operating properly.
User's Guide	Provides in depth information on the HYSYS interface and architecture. HYSYS Utilities are also covered in this manual.
Simulation Basis	Contains all information relating to the available HYSYS fluid packages and components. This includes information on the Oil Manager, Hypotheticals, Reactions as well as a thermodynamics reference section.
Steady State Modeling	Steady state operation of HYSYS unit operations is covered in depth in this manual.
Dynamic Modeling	This manual contains information on building and running HYSYS simulations in Dynamic mode. Dynamic theory, tools, dynamic functioning of the unit operations as well as controls theory are covered. This manual is only included with the HYSYS.Plant document set.
Customization Guide	Details the many customization tools available in HYSYS. Information on enhancing the functionality of HYSYS by either using third-party tools to programmatically run HYSYS (Automation), or by the addition of user-defined Extensions is covered. Other topics include the current internally extensible tools available in HYSYS: the User Unit Operation and User Variables as well as comprehensive instruction on using the HYSYS View Editor.
Tutorials	Provides step-by-step instructions for building some industry-specific simulation examples.
Applications	Contains a more advanced set of example problems. Note that before you use this manual, you should have a good working knowledge of HYSYS. The Applications examples do not provide many of the basic instructions at the level of detail given in the Tutorials manual.
Quick Reference	Provides quick access to basic information regarding all common HYSYS features and commands.

Contact Hyprotech for information on HYSYS training courses.

If you are new to HYSYS, you may want to begin by completing one or more of the HYSYS tutorials, which give the step-by-step instructions needed to build a simulation case. If you have some HYSYS experience, but would still like to work through some more advanced sample problems, refer to the HYSYS Applications.

Since HYSYS is totally interactive, it provides virtually unlimited flexibility in solving any simulation problem. Keep in mind that the approach used in solving each example problem presented in the HYSYS documentation may only be one of the many possible methods. You should feel free to explore other alternatives.

Viewing the Online Documentation

HYSYS electronic documentation is viewed using **Adobe Acrobat Reader®**, which is included on the Documentation CD-ROM. Install Acrobat Reader 4.0 on your computer following the instructions on the CD-ROM insert card. Once installed, you can view the electronic documentation either directly from the CD-ROM, or you can copy the **Doc** folder (containing all the electronic documentation files) and the file named **menu.pdf** to your hard drive before viewing the files.

Manoeuvre through the online documentation using the *bookmarks* on the left of the screen, the navigation buttons in the button bar or using the scroll bars on the side of the view. Blue text indicates an active link to the referenced section or view. Click on that text and Acrobat Reader will jump to that particular section.

Selecting the Search Index

*Ensure that your version of Acrobat Reader has the **Search** plug-in present. This plug-in allows you to add a search index to the search list.*

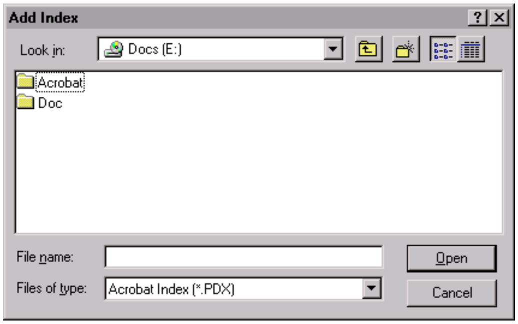
For more information on the search tools available in Acrobat Reader, consult the help files provided with the program.

One of the advantages in using the HYSYS Documentation CD is the ability to do power searching using the Acrobat search tools. The Acrobat **Search** command allows you to perform full text searches of PDF documents that have been indexed using **Acrobat Catolog®**.

To attach the index file to Acrobat Reader 4.0, use the following procedure:

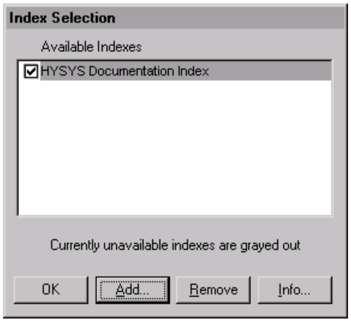
1. Open the **Index Selection** view by selecting **Edit-Search-Select Indexes** from the menu.
2. Click the ***Add*** button. This will open the **Add Index** view.
3. Ensure that the **Look in** field is currently set to your CD-ROM drive label. There should be two directories visible from the root directory: Acrobat and Doc.

Figure 2



4. Open the Doc directory. Inside it you should find the **Index.pdx** file. Select it and click the ***Open*** button.

Figure 3



5. The Index Selection view should display the available indexes that can be attached. Select the index name and then click the ***OK*** button. You may now begin making use of the Acrobat **Search** command.

Using the Search Command

The Acrobat **Search** command allows you to perform a search on PDF documents. You can search for a simple word or phrase, or you can expand your search by using wild-card characters and operators.

To search an index, first select the indexes to search and define a *search query.* A *s*earch query is an expression made up of text and other items to define the information you want to define. Next, select the documents to review from those returned by the search, and then view the occurrences of the search term within the document you selected

To perform a full-text search do the following:

1. Choose **Edit-Search-Query** from the menu.
2. Type the text you want to search for in the Find Results Containing Text box.
3. Click ***Search***. The Search dialog box is hidden, and documents that match your search query are listed in the Search Results window in order of relevancy.
4. Double-click a document that seems likely to contain the relevant information, probably the first document in the list. The document opens on the first match for the text you typed.
5. Click the ***Search Next*** button or ***Search Previous*** button to go to other matches in the document. Or choose another document to view.

Other Acrobat Reader features include a zoom-in tool in the button bar, which allows you to magnify the text you are reading. If you wish, you may print pages or chapters of the online documentation using the **File-Print** command under the menu.

Conventions used in the Manuals

The following section lists a number of conventions used throughout the documentation.

Keywords for Mouse Actions

As you work through various procedures in the manuals, you will be given instructions on performing specific functions or commands. Instead of repeating certain phrases for mouse instructions, keywords are used to imply a longer instructional phrase:

These are the normal (default) settings for the mouse, but you can change the positions of the left- and right-buttons.

Keywords	Action
Point	Move the mouse pointer to position it over an item. For example, point to an item to see its Tool Tip.
Click	Position the mouse pointer over the item, and rapidly press and release the left mouse button. For example, click ***Close*** button to close the current window.
Right-Click	As for click, but use the right mouse button. For example, right-click an object to display the Object Inspection menu.

Keywords	Action
Double-Click	Position the mouse pointer over the item, then rapidly press and release the left mouse button twice. For example, double-click the ***HYSYS*** icon to launch the program.
Drag	Position the mouse pointer over the item, press and hold the left mouse button, move the mouse while the mouse button is down, and then release the mouse button. For example, you drag items in the current window, to move them.
Tool Tip	Whenever you pass the mouse pointer over certain objects, such as tool bar icons and flowsheet objects, a Tool Tip will be displayed. It will contain a brief description of the action that will occur if you click on that button or details relating to the object.

A number of text formatting conventions are also used throughout the manuals:

Note that blank spaces are acceptable in the names of streams and unit operations.

Format	Example
When you are asked to access a HYSYS menu command, the command is identified by bold lettering.	'Select **File-Save** from the menu to save your case.'
When you are asked to select a HYSYS button, the button is identified by bold, italicized lettering.	'Click the ***Close*** button to close the current view.'
When you are asked to select a key or key combination to perform a certain function, keyboard commands are identified by words in bolded small capitals (small caps).	'Press the **F1** key on the keyboard to open the context sensitive help.'
The name of a HYSYS view (also know as a property view or window) is indicated by bold lettering.	'Selecting this command opens the **Session Preferences** view.'
The names of *pages* and *tabs* on various views are identified in bold lettering.	'Click **Composition** page on the **Worksheet** tab to see all the stream composition information.'
The name of *radio* buttons, *check boxes* and *cells* are identified by bold lettering.	'Click the **Ignored** check box to ignore this operation.'
Material and energy stream names are identified by bold lettering.	**Column Feed, Condenser Duty**
Unit operation names are identified by bold lettering.	**Inlet Separator, Atmospheric Tower**
When you are asked to provide keyboard input, it will be indicated by bold lettering.	'Type **100** in the cell to define the stream temperature.'

Bullets and Numbering

Bulleted and numbered lists will be used extensively throughout the manuals. Numbered lists are used to break down a procedure into steps, for example:

1. Select the Name cell.
2. Type a name for the operation.
3. Press **ENTER** to accept the name.

Bulleted lists are used to identify alternative steps within a procedure, or for simply listing like objects. A sample procedure that utilizes bullets is:

1. Move to the Name cell by doing one of the following:
   - Select the Name cell
   - Press **ALT N**
2. Type a name for the operation.
   - Press **ENTER** to accept the name.

Notice the two alternatives for completing Step 1 are indented to indicate their sequence in the overall procedure.

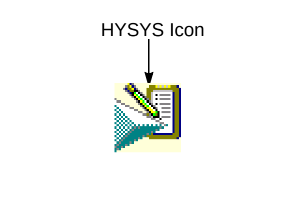
A bulleted list of like objects might describe the various groups on a particular view. For example, the **Options** page of the **Simulation** tab on the Session Preferences view has three groups, namely:

- General Options
- Errors
- Column Options

Callouts

A callout is a label and arrow that describes or identifies an object. An example callout describing a graphic is shown below.

Figure 4



Annotations

Annotation text appears in the outside page margin.

Text appearing in the outside margin of the page supplies you with additional or summary information about the adjacent graphic or paragraph. An example is shown to the left.

Shaded Text Boxes

A shaded text box provides you with important information regarding HYSYS' behaviour, or general messages applying to the manual. Examples include:

> The resultant temperature of the mixed streams may be quite different than those of the feed streams, due to mixing effects.

> Before proceeding, you should have read the introductory section which precedes the example problems in this manual.

The use of many of these conventions will become more apparent as you progress through the manuals.

HYSYS Hot Keys

File

Create New Case	CTRL+N
Open Case	CTRL+O
Save Current Case	CTRL+S
Save As...	CTRL+SHIFT+S
Close Current Case	CTRL+Z
Exit HYSYS	ALT+F4

Simulation

Go to Basis Manager	CTRL+B
Leave Current Environment (Return to Previous)	CTRL+L
Main Properties	CTRL+M
Access Optimizer	F5
Toggle Steady-State/Dynamic Modes	F7
Toggle Hold/Go Calculations	F8
Access Integrator	CTRL+I
Start/Stop Integrator	F9
Stop Calculations	CTRL+BREAK

Flowsheet

Add Material Stream	F11
Add Operation	F12
Access Object Navigator	F3
Show/Hide Object Palette	F4
Composition View (from Workbook)	CTRL+K

Tools

Access Workbooks	CTRL+W
Access PFDs	CTRL+P
Toggle Move/Attach (PFD)	CTRL
Access Utilities	CTRL+U
Access Reports	CTRL+R
Access DataBook	CTRL+D
Access Controller FacePlates	CTRL+F
Access Help	F1

Column

Go to Column Runner (SubFlowsheet)	CTRL+T
Stop Column Solver	CTRL+BREAK

Window

Close Active Window	CTRL+F4
Tile Windows	SHIFT+F4
Go to Next Window	CTRL+F6 or CTRL+TAB
Go to Previous Window	CTRL+SHIFT+F6 or CTRL+SHIFT+TAB

Editing/General

Access Edit Bar	F2
Access Pull-Down Menus	F10 or ALT
Go to Next Page Tab	CTRL+SHIFT+N
Go to Previous Page Tab	CTRL+SHIFT+P
Cut	CTRL+X
Copy	CTRL+C
Paste	CTRL+V

1 Dynamic Theory

Dynamic simulation can help you to better design, optimize, and operate your chemical process or refining plant. Chemical plants are never truly at steady state. Feed and environmental disturbances, heat exchanger fouling, and catalytic degradation continuously upset the conditions of a smooth running process. The transient behaviour of the process system is best studied using a dynamic simulation tool like HYSYS.

The design and optimization of a chemical process involves the study of both steady state and dynamic behaviour. Steady state models can perform steady state energy and material balances and evaluate different plant scenarios. The design engineer can use steady state simulation to optimize the process by reducing capital and equipment costs while maximizing production.

With dynamic simulation, you can confirm that the plant can produce the desired product in a manner that is safe and easy to operate. By defining detailed equipment specifications in the dynamic simulation, you can verify that the equipment will function as expected in an actual plant situation. Off-line dynamic simulation can optimize controller design without adversely affecting the profitability or safety of the plant. You can design and test a variety of control strategies before choosing one that may be suitable for implementation. You can examine the dynamic response to system disturbances and optimize the tuning of controllers. Dynamic analysis provides feedback and improves the steady state model by identifying specific areas in a plant that may have difficulty achieving the steady state objectives.

In HYSYS, the dynamic analysis of a process system can provide insight into understanding it that is not possible with steady state modelling. With dynamic simulation you can investigate:

- Process optimization
- Controller optimization
- Safety evaluation
- Transitions between operating conditions
- Startup/Shutdown conditions

The HYSYS dynamic model shares the same physical property packages as the steady state model. The dynamic model simulates the thermal, equilibrium and reactive behaviour of the chemical system in a similar way to the steady state model.

On the other hand, the dynamic model uses a different set of conservation equations which account for changes occurring over time. The equations for material, energy, and composition balances include an additional "accumulation" term which is differentiated with respect to time. Non-linear differential equations can be formulated to approximate the conservation principles; however, an analytical solution method does not exist.

Therefore, numerical integration is used to determine the process behaviour at distinct time steps. The smaller the time step, the more closely the calculated solution will match the analytic solution. However, this gain in rigour is offset by the additional calculation time required to simulate the same amount of elapsed real time. A reasonable compromise may be achieved by using the largest possible step size, while maintaining an acceptable degree of accuracy without becoming unstable.

The HYSYS dynamic simulation package has the capacity to reach a wider audience by offering the following features demanded by industry:

- **Accuracy**. The HYSYS dynamic model provides accurate results based on rigorous equilibrium, reaction, unit operations and controller models. You must be able to trust the results if the dynamic tool is to be useful at all.
- **Ease of Use**. The HYSYS dynamic package uses the same intuitive and interactive graphical environment as the HYSYS steady state model. Streams and unit operations in the flowsheet can be added to the dynamic simulation environment as easily as in steady state. All flowsheet information from a steady state simulation case transfers easily to the dynamic simulation environment.
- **Speed**. The dynamic modelling options in HYSYS have been developed to provide a balance between accuracy and speed. HYSYS uses a fixed step size implicit Euler method. Volume, energy, and composition balances are solved at different frequencies. Volume (Pressure-Flow) balances are defaulted to solve at every time step, whereas energy and composition balances are defaulted to solve at every 2nd and 10th time step, respectively. This solution method allows HYSYS to perform quick, accurate and stable calculations in your simulation case.
- **Detailed Design**. You can provide specific rating details for each piece of equipment in the plant and confirm that the specified equipment can achieve desired product specs and quality. Rating information includes the equipment size, geometry, nozzle placement, and position relative to the ground. A comprehensive holdup model calculates levels, heat loss, static head contributions and product compositions based on the rating information of each piece of equipment.

- **Realism**. A new level of realism with regards to material flow within the simulation is achieved with the **Pressure Flow** solver. With the **Pressure Flow** option, the flow rate through any unit operation depends on the pressures of the surrounding pieces of equipment. Material flow through an actual plant can be more accurately modelled using the **Pressure Flow** solver.
- **Customizable**. The HYSYS dynamic model is customizable. Many organizations have proprietary information that they wish to integrate into their commercial simulator platform. HYSYS allows you to add your own OLE modules to the HYSYS dynamic simulation environment.

1.1 General Concepts

Mathematical Model Classification

Distributed and Lumped Models

Most chemical engineering systems have thermal or component concentration gradients in three dimensions (x,y,z) as well as in time. This is known as a distributed system. If you were to characterize such a system mathematically, you would obtain a set of partial differential equations (PDEs).

If the x, y and z gradients are ignored, the system is "lumped", and all physical properties are considered to be equal in space. Only the time gradients are considered in such an analysis. This consideration allows for the process to be described using ordinary differential equations (ODE's) which is much less rigorous than PDEs, thereby saving calculation time. For most instances, the lumped method will give a solution which is a reasonable approximation of the distributed model solution.

HYSYS uses lumped models for all of the unit operations. For instance, in the development of the equations describing the separator operation, it is assumed that there are no thermal or concentration gradients present in a single phase. In other words, the temperature and composition of each phase are the same throughout the entire separator.

Note that by definition, the PFR has thermal and concentration gradients with respect to the length of the vessel. In the solution algorithm, the PFR reactor is subdivided into several subvolumes which are considered to be lumped; that is, the reaction rate, temperature and compositions are constant through each subvolume, varying only with time. In essence, therefore, the PFR model, though inherently distributed (with respect to the length of the vessel), still uses a lumped analysis to obtain the solution.

Linear and Non-Linear Systems

A linear first-order Ordinary Differential Equation (ODE) can be described as follows:

$$\tau\frac{dY}{dt} + Y = Kf(u) \tag{1.1}$$

In a non-linear equation, the process variable Y may appear as a power, exponential, or is not independent of other process variables. Here are two examples:

$$\tau\frac{dY}{dt} + Y^3 = Kf(u) \tag{1.2}$$

$$\tau\frac{dY}{dt} + YY_2 = Kf(u) \tag{1.3}$$

The great majority of chemical engineering processes occurring in nature are nonlinear. Nonlinearity may arise from equations describing equilibrium behaviour, fluid flow behaviour, or reaction rates of chemical systems. While a linear system of equations may be solved analytically using matrix algebra, the solution to a non-linear set of equations usually requires the aid of a computer.

Conservation Relationships

Material Balance

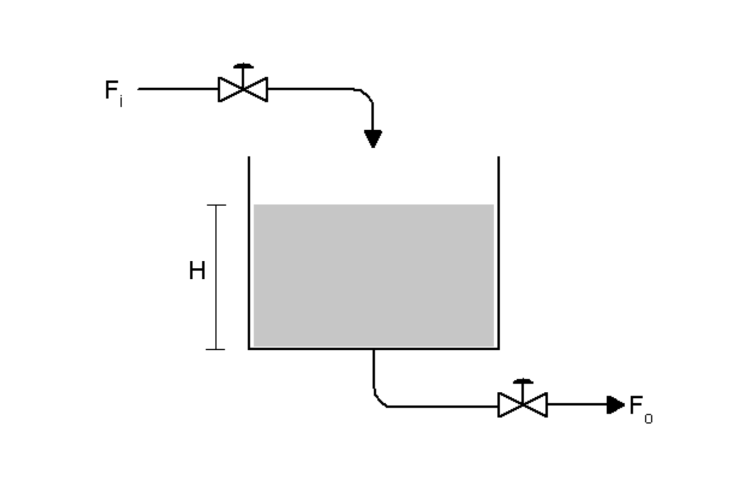
The conservation relationships are the basis of mathematical modelling in HYSYS. The dynamic mass, component, and energy balances that are derived in the following section are similar to the steady-state balances with the exception of the accumulation term in the dynamic balance. It is the accumulation term which allows the output variables from the system to vary with time.

The conservation of mass is maintained in the following general relation:

Rate of accumulation of mass = mass flow into system - mass flow out of system (1.4)

For the simple case of a perfectly mixed tank with a single component feed, the mass balance would be as follows:

Figure 1.1



$$\frac{d(\rho_o V)}{dt} = F_i \rho_i - F_o \rho_o \tag{1.5}$$

where: F_i = *the flowrate of the feed entering the tank*

ρ_i = *the density of the feed entering the tank*

F_o = *the flowrate of the product exiting the tank*

ρ_o = *the density of the product exiting the tank*

V = *the volume of the fluid in the tank*

Component Balance

Component balances can be written as follows:

$$\begin{aligned}&\textit{Rate of accumulation of component } j = \\ &\quad \textit{Flow of component } j \textit{ into system} \\ &- \textit{Flow of component } j \textit{ out of system} \\ &+ \textit{Rate of formation of component } j \textit{ by reaction}\end{aligned} \tag{1.6}$$

Flow into or out of the system can be convective (bulk flow) and/or molecular (diffusion). While convective flow contributes to the majority of the flow into and out of a system, diffusive flow may become significant if there is a high interfacial area to volume ratio for a particular phase.

For a multi-component feed for a perfectly mixed tank, the balance for component *j* would be as follows:

$$\frac{d(C_{jo}V)}{dt} = F_i C_{ji} - F_o C_{jo} + R_j V \tag{1.7}$$

where: C_{ji} *= the concentration of j in the inlet stream*

C_{jo} *= the concentration of j in the outlet stream*

Rj *= the reaction of rate of the generation of component j*

For a system with NC components, there are NC component balances. The total mass balance and component balances are not independent; in general, you would write the mass balance and NC-1 component balances.

Energy Balance

The Energy balance is as follows:

$$\begin{aligned}&\textit{Rate of accumulation of total energy} =\\&\quad \textit{Flow of total energy into system}\\&- \textit{Flow of total energy out of system}\\&+ \textit{Heat added to system across its boundary}\\&+ \textit{Heat generated by reaction}\\&- \textit{Work done by system on surroundings}\end{aligned} \tag{1.8}$$

The flow of energy into or out of the system is by convection or conduction. Heat added to the system across its boundary is by conduction or radiation.

For a CSTR with heat removal, the following general equation applies:

$$\frac{d}{dt}[(u+k+\phi)V] = F_i\rho_i(u_i+k_i+\phi_i) - F_o\rho_o(u_o+k_o+\phi_o) + Q + Q_r - (w + F_oP_o - F_iP_i) \tag{1.9}$$

where: u = *Internal energy (energy per unit mass)*

k = *Kinetic energy (energy per unit mass)*

ϕ = *Potential energy (energy per unit mass)*

V = *the volume of the fluid*

w = *Shaft work done by system (energy per time)*

P_o = *Vessel pressure*

P_i = *Pressure of feed stream*

Q = *Heat added across boundary*

Q_r = *Heat generated by reaction:* $DH_{rxn}r_A$

Several simplifying assumptions can usually be made:

- The potential energy can almost always be ignored; the inlet and outlet elevations are roughly equal.
- The inlet and outlet velocities are not high, therefore kinetic energy terms are negligible.
- If there is no shaft work (no pump), *w*=0.

The general energy balance for a 2-phase system is as follows:

$$\frac{d}{dt}[\rho_v V_v H + \rho_l V_l h] = F_i \rho_i h_i - F_l \rho_l h - F_v \rho_v H + Q + Q_r \qquad (1.10)$$

Solution Method

Implicit Euler Method

Y_{n+1} may be calculated analytically calculated to equal:

$$Y_{n+1} = Y_n + \int_{t_n}^{t_{n+1}} f(Y)dt \qquad (1.11)$$

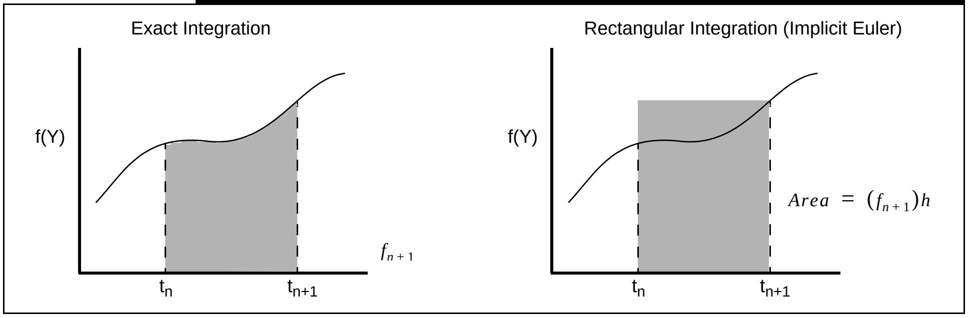
where: $\frac{dY}{dt} = f(Y)$

Ordinary differential equations may be solved using the implicit Euler method. The implicit Euler method is simply an approximation of Y_{n+1} using rectangular integration. Graphically, a line of slope zero and length h (the step size) is extended from t_n to t_{n+1} on a $f(Y)$ versus time plot. The area under the curve is approximated by a rectangle of length h and height $f_{n+1}(Y_{n+1})$:

$$Y_{n+1} = Y_n + hf_{n+1}(Y_{n+1}) \qquad (1.12)$$

Figure 1.2 shows the integration of $f(Y)$ over time step, h, using exact integration and the implicit Euler approximation:

Figure 1.2



The implicit Euler method handles stiff systems well. This is an implicit method because information is required at time t_{n+1}. Integration parameters such as the integration time step can be specified in the **Integrator** view from the **Simulation** menu in HYSYS. The integration time step can be adjusted to increase the speed or stability of the system.

Integration Strategy

In HYSYS.Plant, dynamic calculations are performed at three different frequencies:

- Volume (Pressure-flow)
- Energy
- Composition

These relations are not solved simultaneously at every time step. This would be computationally expensive. The compromise is to solve the balances at different time step frequencies. The default solution frequencies, which are multiples of the integration time step, are 1,2, and 10 for the pressure flow equations, energy, and composition balances, respectively. That is, pressure flow equations are solved at every time step while composition balances are solved at every 10th time step. Since composition tends to change much more gradually than the pressure, flow, or energy in a system, the equations associated with composition can be solved less frequently. An approximate flash is used for each pressure flow integration time step. A rigorous flash is performed at every composition integration time step.

1.2 Holdup Model

Dynamic behaviour arises from the fact that many pieces of plant equipment have some sort of material inventory or holdup. A holdup model is necessary because changes in the composition, temperature, pressure or flow in an inlet stream to a vessel with volume (holdup) are not immediately seen in the exit stream. The model predicts how the holdup and exit streams of a piece of equipment respond to input changes to the holdup over time.

In some cases, the holdup model corresponds directly with a single piece of equipment in HYSYS.Plant. For example, a separator is considered a single holdup. In other cases, there are numerous holdups within a single piece of equipment. In the case of a distillation column, each tray can be considered a single holdup. Heat exchangers can also be discretized into zones with each zone being a set of holdups.

Calculations included in the holdup model are:

- Material and Energy Accumulation
- Thermodynamic Equilibrium
- Heat Transfer
- Chemical Reaction

The new holdup model offers certain advantages over the previous HYSYS dynamic model:

1. An adiabatic PH flash calculation replaces the bubble point algorithm used in the previous holdup model. Adiabatic flashes also allow for more accurate calculations of vapour composition and pressure effects in the vapour holdup.
2. Flash efficiencies can be specified which allows for the modelling of non-equilibrium behaviour between the feed phases of the holdup.
3. The placement of feed and product nozzles on the equipment has physical meaning in relation to the holdup. For instance, if the vapour product nozzle is placed below the liquid level in a separator, only liquid will exit from the nozzle.

1.2.1 Assumptions of Holdup Model

There are several underlying assumptions that are considered in the calculations in the holdup model:

1. Each phase is assumed to be well mixed.
2. Mass and heat transfer occur between feeds to the holdup and material already in the holdup.
3. Mass and heat transfer occurs between phases in the holdup.

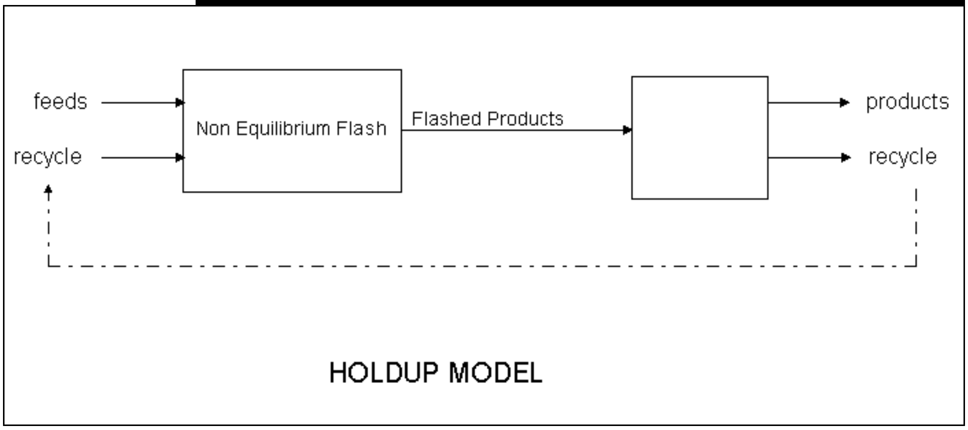
1.2.2 Accumulation

The lagged response that is observed in any unit operation is the result of the accumulation of material, energy, or composition in the holdup. In order to predict how the holdup conditions change over time, a recycle stream is added alongside the feed streams. For instance, the material accumulation in a holdup can be calculated from:

$$\begin{aligned}
&\textit{Material accumulation}_{\text{new}} = \textit{material flow into system}\\
&\quad + \textit{material accumulation}_{\text{old}}\ \textit{(recycle stream)} \qquad (1.13)\\
&\quad - \textit{material flow out of system}
\end{aligned}$$

The recycle stream is not a physical stream in the unit operation. Rather, it is used to introduce a lagged response in the output. Essentially, the recycle stream represents the material already existing in the piece of equipment. It becomes apparent that a greater amount material in the holdup means a larger recycle stream and thus, a greater lagged response in the output.

Figure 1.3



The holdup model is used to calculate material, energy, and composition accumulation. Material accumulation is defaulted to calculate at every integration time step. The energy of the holdup is defaulted to calculate at every 2nd time step. The composition of the holdup is defaulted to calculate at every 10th time step.

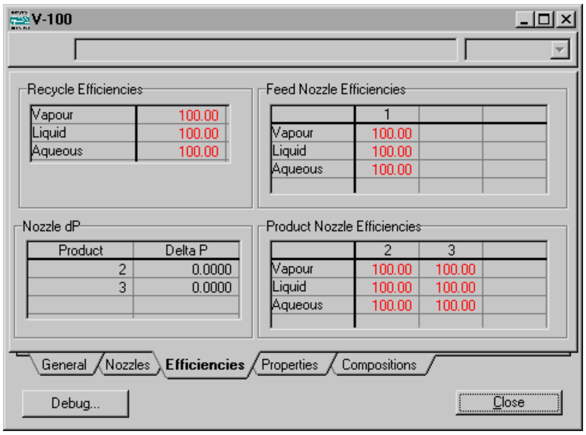
1.2.3 Non-Equilibrium Flash

As material enters a holdup, the liquid and vapour feeds may associate in different proportions with the existing material already in the holdup. For instance, a separator's vapour and liquid feeds may enter the column differently. It is very likely that the liquid feed will mix well with the liquid already in the holdup. The vapour feed may not mix as well with the existing material in the vessel since the residence time of the vapour holdup is much smaller than the liquid. If the feed nozzle is situated close to the vapour product nozzle, it is possible that even less mixing will occur. In the physical world, the extent of mixing of feeds with a holdup depends on the placement of the feed nozzles, the amount of holdup, and the geometry of the piece of equipment.

Efficiencies

In HYSYS, you can indirectly specify the amount of mixing that occurs between the feed phases and the existing holdup using feed, recycle, and product efficiencies. These feed efficiency parameters may be specified in the unit operation's **Holdup** page under the **Dynamics** tab in HYSYS.Plant.

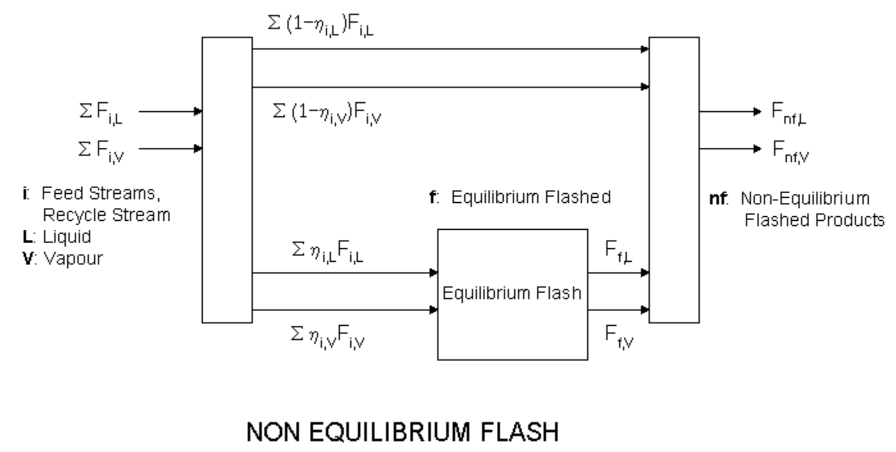
Figure 1.4



Essentially, the efficiency represents how close the feed comes to equilibrium with the other feeds. If all feed phases enter the holdup at 100% efficiencies, the holdup composition, temperature, and phase fraction will eventually match the flashed feeds' stream conditions.

A flash efficiency can be specified for each phase of any stream entering the holdup. A conceptual diagram of the non-equilibrium flash is shown for a two phase system in **Figure 1.5**:

Figure 1.5



As shown, the flash efficiency, η, is the fraction of feed stream that participates in the rigorous flash. If the efficiency is specified as 1, the entire stream participates in the flash; if the efficiency is 0, the entire stream bypasses the flash and is mixed with the product stream.

The recycle stream and any streams entering the holdup participates in the flash. You can specify the flash efficiency for each phase of the recycle stream and any feed entering the holdup. The flash efficiency may also be specified for each phase of any product streams leaving the holdup.

> Note: Product flash efficiencies are only used by the holdup model when reverse flow occurs in the product flow nozzles. In such cases, the product nozzle effectively becomes a feed nozzle and uses the product flash efficiencies provided by you, the user.

The default efficiencies for the feed, product, and recycle streams is 1. The flash efficiencies should be changed if it is observed that most of the vapour feed to the holdup condenses in the holdup. This could adversely affect the pressure of the holdup and consequently the entering and exiting stream flow rates.

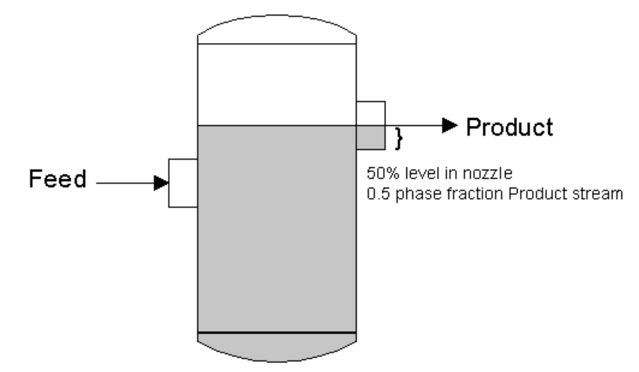
For instance, a water system which is heated by pure steam (no inerts) may encounter problems if the steam feed efficiency is specified as 1. If the holdup material is significantly larger than the steam flow, all the steam will condense and the holdup temperature will increase, accordingly. Because no vapour effectively enters the holdup, the pressure will collapse if the vapour space in the holdup is significant. In the physical world, a small amount of steam flow will condense in the water system. The great majority of steam will bubble through the steam and maintain the pressure in the vessel. This should be modelled in HYSYS.Plant by specifying the steam feed efficiency to be less than 1.

Nozzles

In HYSYS.Plant, you may specify the feed and product nozzle locations and diameters. These nozzle placement parameters may be specified in the unit operation's **Nozzles** page under the **Rating** tab in the operation's view.

The placement of feed and product nozzles on the equipment has physical meaning in relation to the holdup. The exit stream's composition depends partially on the exit stream nozzle's location 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 will draw material from the liquid holdup. If the product nozzle is located above the liquid level, the exit stream will draw 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.

Figure 1.6



Static Head Contributions

When the **Static Head Contributions** check box is activated on the **Options** tab of the **Integrator** view, HYSYS calculates static head using the following contributions:

- Levels inside separators, tray sections, etc.
- Elevation differences between connected equipment

Including static head contributions in the modelling of pressure-flow dynamics is an option in HYSYS.

For unit operations that have *negligible* holdup, such as the valve operation, HYSYS incorporates only the concept of nozzles. There is no static head contributions for levels, unless the feed and product nozzles are specified at different elevations. You can specify the elevation of both the feed and product nozzles. If there is a difference in elevation between the feed and product nozzles, HYSYS uses this value to calculate the static head contributions. It is recommended that static head contributions not be modelled in these unit operations in this way since this is not a realistic situation. Static head can be better modelled in these unit operations by relocating the entire piece of equipment.

Static head is important in vessels with levels. For instance, consider a vertical separator unit operation that has a current liquid level of 50%. The static head contribution of the liquid holdup will make the pressure at the liquid outlet nozzle higher than that at the vapour outlet nozzle. Nozzle location will also become a factor. The pressure-flow relationship for the separator will be different for a feed nozzle which is below the current liquid holdup level as opposed to a feed which is entering in the vapour region of the unit.

It is important to note that exit stream pressures from a unit operation are calculated at the exit nozzle locations on the piece of equipment and not the inlet nozzle locations of the next piece of equipment.

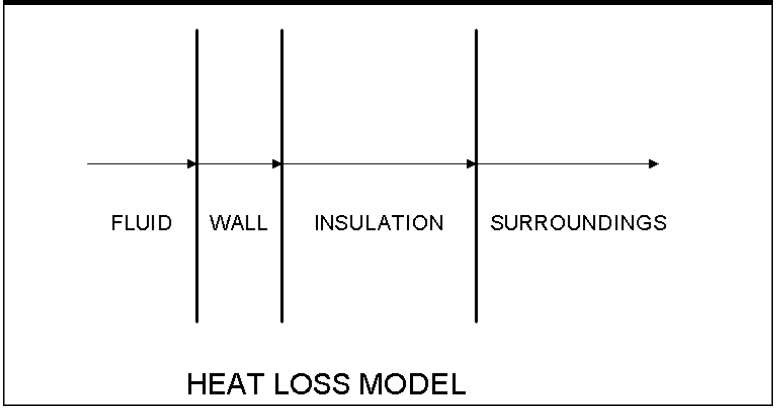
1.2.4 Heat Loss Model

The heat loss experienced by any pieces of plant equipment is considered by the holdup model in HYSYS. The heat loss model influences the holdup by contributing an extra term to the energy balance equation.

Energy Balances

Heat is lost (or gained) from the holdup fluid through the wall and insulation to the surroundings.

Figure 1.7



There are several underlying assumptions that are considered during a heat loss calculation:

- There is heat capacity associated with the wall and insulation housing the fluid.
- The temperature across the wall and insulation is assumed to be constant. (lumped parameter analysis)
- The heat transfer coefficient between the holdup and the wall is assumed to be same for the vapour and liquid.

A balance can be performed across the wall:

$$\frac{d}{dt}[Ax_{wall}Cp_{wall}T_{wall}] = h_{(fluid,\,wall)}A(T_{fluid} - T_{wall}) - \frac{k_{ins}}{x_{ins}}A(T_{wall} - T_{ins}) \tag{1.14}$$

The balance across the insulation is:

$$\frac{d}{dt}\left[Ax_{ins}Cp_{ins}\left(\frac{T_{wall} + T_{ins}}{2}\right)\right] = \frac{k_{ins}}{x_{ins}}A(T_{wall} - T_{ins}) + h_{(ins,\,surr)}A(T_{ins} - T_{surr}) \tag{1.15}$$

where: A = Heat transfer area

x = thickness

Cp = Heat capacity

T = temperature

k = thermal conductivity

h = heat transfer coefficient

As shown, both the insulation and wall can store heat. The heat loss term that is accounted for in the energy balance around the holdup is $h_{(fluid,\ wall)}A(T_{fluid} - T_{wall})$. If T_{fluid} is greater than T_{wall}, the heat will be lost to the surroundings. If T_{fluid} is less than T_{wall}, the heat will be gained from the surroundings.

Change in Vessel Level

If the vessel level changes, a part of the wall and insulation that was associated with one phase will become part of another. The temperature of the incremental piece of the wall and the bulk of the wall will be equilibrated by the simple averaging:

$$T_{new} = \frac{T_1 A + T_2 \delta A}{A + \delta A} \tag{1.16}$$

where: A = Heat transfer area of encroaching phase

T_1 = *temperature of encroaching phase*

T_2 = *temperature of displaced phase*

Heat Loss Parameters

The heat loss parameters can be specified for most unit operations in the **Heat Loss** page under the **Rating** tab. You may choose to neglect the heat loss calculation in the energy balance by selecting the **None** radio button.

Two heat loss models are available to you: **Simple** and **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.Plant. The heat loss is calculated using:

$$Q = UA(T_f - T_{amb}) \tag{1.17}$$

Detailed Model

The Detailed model allows you to specify more detailed heat transfer parameters. There are three radio buttons in the **Heat Loss Parameters** group as described in the table below.

Radio Button	Description
Temperature Profile	Displays the temperatures of the: • fluid • wall • insulation • surroundings
Conduction	Displays the conductive properties of the **wall** and **insulation**. The following properties can be specified by you: • Conductivity of material • Thickness of material • Heat capacity of material • Density of material **Equation (1.14)** and **(1.15)** demonstrate how the parameters are used by the heat loss model.
Convection	Displays the convective heat transfer coefficients for heat transfer within the holdup and heat transfer occurring from the outside the holdup to the surroundings.

1.2.5 Chemical Reactions

Chemical reactions that occur in plant equipment are considered by the holdup model in HYSYS.Plant. Reaction sets may be specified in the **Results** page of the **Reactions** tab.

The holdup model is able to calculate the chemical equilibria and reactions that occur in the holdup. In a holdup, chemical reactions may be modelled by one of four mechanisms:

*For more information on how reaction sets can be created and used within the simulation, see **Chapter 4 - Reactions** in the **Simulation Basis** manual.*

- Reactions handled inside thermophysical property packages
- Extent of reaction model
- Kinetic model
- Equilibrium model

1.2.6 Related Calculations

There are calculations which are not handled by the holdup model itself but may impact the holdup calculations. The following calculations require information and are solved in conjunction with the holdup model:

Vessel Level Calculations

The vessel level can be calculated from the vessel geometry, the molar holdup and the density for each liquid phase.

Vessel Pressure

The vessel pressure is a function of the vessel volume and the stream conditions of the feed, product, and the holdup. The pressure in the holdup is calculated using a volume balance equation. Holdup pressures are calculated simultaneously across the flowsheet.

Tray Hydraulics

Tray Hydraulics determines the rate from which liquid leaves the tray, and hence, the holdup and the pressure drop across the tray. The Francis Weir equation is used to determine the liquid flow based on the liquid level in the tray and the tray geometry.

1.2.7 Advanced Holdup Properties

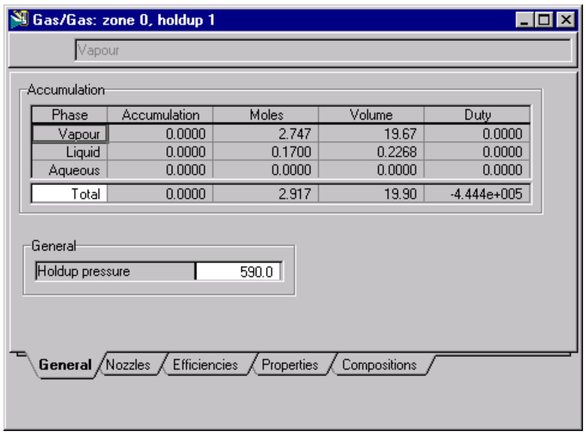
*Click the right mouse button anywhere in the view to bring up the **Open Page** button. Clicking this button displays the information on the general tab in a separate window*

Located on each **Holdup** pages found on the **Dynamics** tab of the unit operation property view there is an **Advanced** button. This button accesses a view that provides more detailed information about the holdup of that unit operation.

General Tab

This tab provides the same information as shown in the **Holdup** page of the **Dynamics** tab. The accumulation, moles, and volume of the holdup are displayed. The holdup pressure is also displayed in this tab.

Figure 1.8

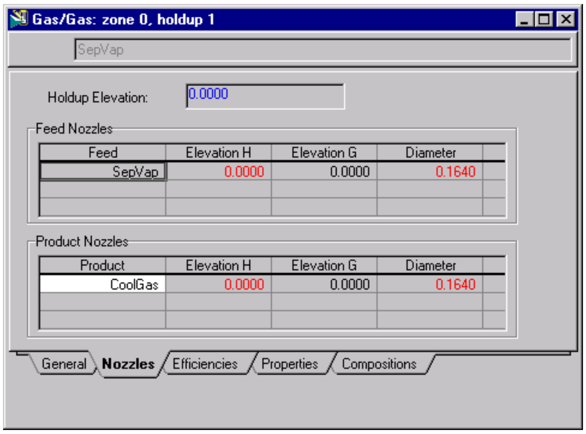


Nozzles Tab

Both the Nozzles tab and Efficiencies tab requires HYSYS.Plant +. Refer to for more information.

This tab displays the same information as shown in the **Nozzles** page of the **Ratings** tab. The nozzle diameters and elevations for each stream attached to the holdup are displayed. This section also displays the holdup elevation which is essentially equal to the base elevation of the piece of equipment relative to the ground. Changes to nozzle parameters can either be made in this tab or in the **Nozzles** page of the **Ratings** tab.

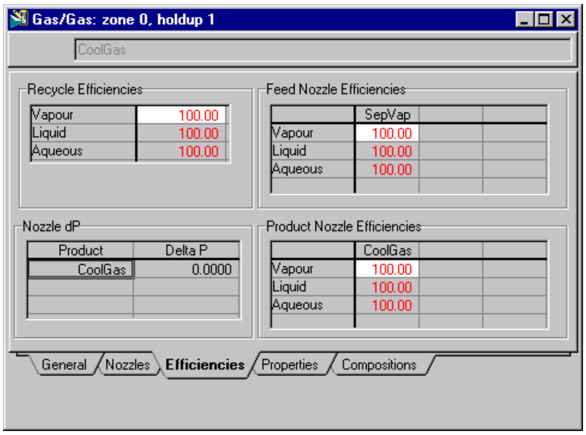
Figure 1.9



Efficiencies Tab

The nozzle efficiencies may be specified in this tab. In HYSYS, you can indirectly specify the amount of mixing that occurs between the feed phases and existing holdup using feed, recycle and product efficiencies.

Figure 1.10



A flash efficiency, η, is the fraction of feed stream that participates in the rigorous flash. If the efficiency is specified as 100, the entire stream participates in the flash; if the efficiency is 0, the entire stream bypasses the flash and is mixed with the product stream.

Nozzle Efficiency	Description
Feed Nozzle Efficiency	The efficiencies of each phase for each feed stream into the holdup can be specified in these cells. These efficiencies are not used by the holdup model if there is flow reversal in the feed streams.
Product Nozzle Efficiency	Product nozzle efficiencies are used only when there is flow reversal in the product streams. In this situation, the product nozzles act as effective feed nozzles.
Recycle Efficiency	Essentially, the recycle stream represents the material already existing in the holdup. Recycle efficiencies represent how much of the material in the holdup participates in the flash.

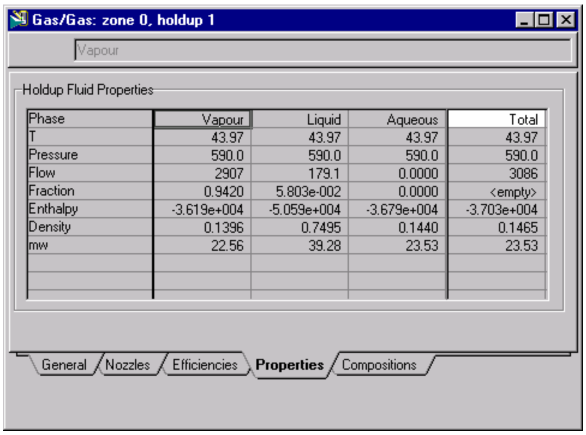
For more information regarding feed, product, and recycle efficiencies, see **Section 1.2.3 - Non-Equilibrium Flash** in this manual.

Properties Tab

The following fluid properties for each phase in the holdup are displayed in the **Properties** tab:

- Temperature
- Pressure
- Flow
- Molar Fraction of the specific phase in the holdup
- Enthalpy
- Density
- Molecular Weight

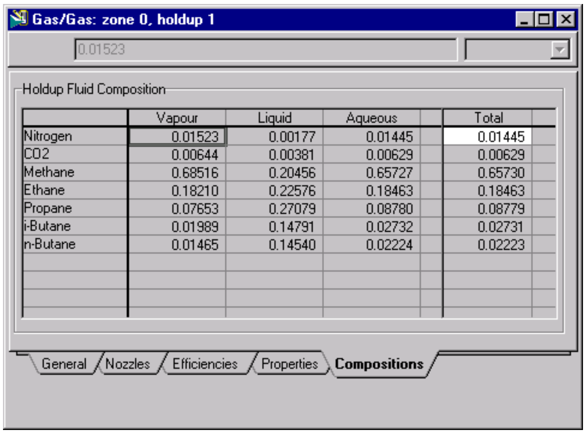
Figure 1.11



Compositions Tab

The compositional molar fractions of each phase in the holdup is displayed in the **Compositions** tab.

Figure 1.12



1.3 Pressure Flow Solver

HYSYS.Plant offers an advanced method of calculating the pressure and flow profile of a simulation case in **Dynamic** mode. Almost every unit operation in the flowsheet can be considered a holdup or carrier of material (pressure) and energy. A network of pressure holdups can therefore be conceived across the entire simulation case. The P-F solver considers the integration of pressure flow balances in the flowsheet. There are two basic equations which define most of the pressure flow network. These equations only contain pressure and flow as variables:

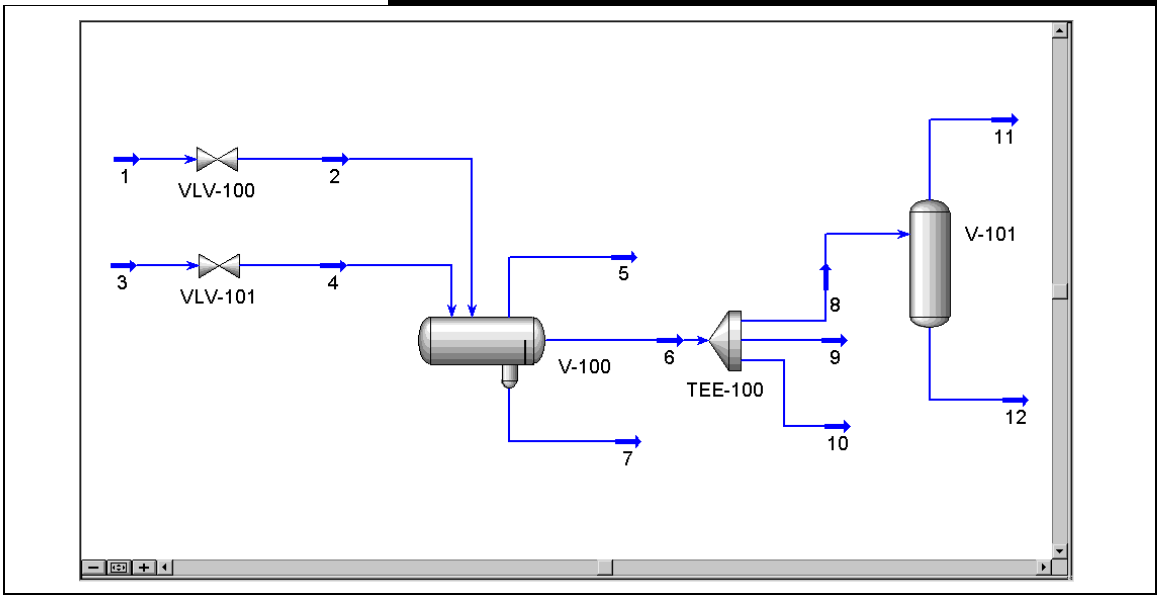
- **Resistance equations** - which define flow between pressure holdups
- **Volume balance equations** - which define the material balance at pressure holdups

The pressure flow balances both require information from and provide information to the holdup model. While the holdup model calculates the accumulation of material, energy, and composition in the holdup, the pressure flow solver equations predict the accumulated pressure of the holdup and flow rates exiting the holdup. The holdup model brings the actual feed and product stream properties to holdup conditions for the volume balance equations using a rigorous or approximate flash. The pressure flow solver returns information essential to the holdup model calculations: the pressure of the holdup or the flow rates of the stream exiting the holdup.

1.3.1 Simultaneous Solution in Pressure Flow Balances

All material streams within HYSYS.Plant can be solved for pressure and flow. All unit operations can be solved for pressure. As an example, consider the following flowsheet. There are 26 variables to solve for in the PF matrix. Twelve material streams contribute 24 variables to the flowsheet. The 2 vessels, V-100 and V-101, contribute 1 variable each. The valve and tee operations are not considered nodes. These unit operations define a pressure flow relation between the inlet and exit streams but rarely are they modelled with any inventory.

Figure 1.13



A pressure-flow matrix is setup which solves the variables required. The matrix consists of: **Volume balance** equations, **Resistance** equations and **Pressure-Flow specifications** input by you. The number of pressure flow specifications that need to be provided by you will be discussed in **Degrees of Freedom Analysis** in Section 1.3.3 - Pressure Flow Specifications.

1.3.2 Basic Pressure Flow Equations

The equations that are discussed in this section define the basis of the pressure flow network.

Volume Balance

For equipment with holdup, an underlying principle is that the physical volume of the vessel, and thus, the volume of material in the vessel at any time remains constant. Therefore, during calculations in dynamics, the change in volume of the material inside the vessel is zero:

$$V = Constant = f(flow, h, P, T) \tag{1.18}$$

$$\frac{dV}{dt} = 0 \tag{1.19}$$

where: V= volume of the vessel

t = time

flow = mass flowrate

h = holdup

P = vessel pressure

T = vessel temperature

As such, a vessel pressure node equation is essentially a volumetric flow balance and can be expressed as follows:

Volume change due to pressure + Volume change due to flows +

Volume change due to temperature + Volume change due to other factors = 0 (1.20)

In the volume balance equation, pressure and flow are the only two variables to be solved in the matrix. All other values in the equation are updated after the matrix solves. Each vessel holdup contributes at least one volume balance equation to the pressure-flow matrix. When sufficient pressure-flow specifications are provided by you, any unknown(s) can be solved whether it be a vessel pressure or one of its flowrates.

The volume balance equation allows you to observe pressure effects in the vapour holdup due to disturbances in the feed. Consider a separator whose feed flow suddenly increases. Assume that the exit streams from the separator are specified by you and are thus, constant. The vessel pressure would increase for 2 reasons:

1. Because the material of the exit streams remain constant, an increase in the vapour feed flow would increase the vapour holdup. An increase in the vapour holdup means that a larger amount of material is compressed into the same vapour volume resulting in a vessel pressure increase.
2. The increase in the liquid level causes the vapour holdup to occupy a smaller volume within the vessel, causing the vessel pressure to rise.

Resistance Equations

Flows exiting from a holdup may be calculated from a volume balance equation, specified by you, or calculated from a resistance equation. In general, the resistance equation calculates flowrates from the pressure differences of the surrounding nodes. HYSYS contains unit operations such as **VALVES** and **HEAT EXCHANGERS** which calculate flowrates using resistance equations. The resistance equations are modelled after turbulent flow equations and have the form:

$$Flow = k\sqrt{\Delta P} \tag{1.21}$$

where: Flow = mass flowrate

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

ΔP = frictional pressure loss which is the pressure drop across the unit operation without static head contributions.

Equation (1.21) is a simplified form of the basic **VALVE** operation equation which uses the valve flow coefficient Cv. The mass flowrate through the valve is a function of the valve flow coefficient and the frictional pressure drop across the valve:

$$Flow = f(Cv, P_1, P_2) \tag{1.22}$$

where: Flow = mass flowrate

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

P_1 *= upstream pressure*

P_2 *= downstream pressure*

As shown, a resistance equation relates the pressures of two nodes and the flow that exists between the nodes. The following unit operations have a resistance equation associated with them.

Unit Operation	Resistance Term
Valve	With a pressure flow specification, you can specify conductance, *Cv*, on the **Specs** page of the **Dynamics** tab.
Pump	The heat flow and pump work define the pressure flow relation of the pump. These parameters can be specified and/or calculated on the **Specs** page of the **Dynamics** tab.
Compressor/Expander	The heat flow and compressor work define the pressure flow relation of the compressor. These parameters can be specified and/or calculated on the **Specs** page of the **Dynamics** tab.
Heater/Cooler/Heat Exchanger/ Air Cooler/LNG	With a pressure flow specification, you can specify the *k-value* on the **Specs** page of the **Dynamics** tab.
Tray Sections, Weir Equation	The Weir equation determines liquid flow rate from the tray as a function of liquid level in the tray. Tray geometry can be specified on the **Sizing** page of the **Ratings** tab.
Tray Sections, K-Value	The K-value is used to determine vapour flow exiting from the tray as a function of the pressure difference between trays. K-values can either be calculated or specified on the **Specs** page of the **Dynamics** tab.

For a more detailed discussion on the individual unit operations and the resistance equations associated with them, see the appropriate unit operation section in the **Dynamic Modelling** guide.

1.3.3 Pressure Flow Specifications

In **Dynamic** mode, you can specify the pressure and/or flow of a material stream in a flowsheet. The pressure flow specifications are made in the **Dynamics** tab of the **Material Stream** property view. In order to satisfy the degrees of freedom of the pressure-flow matrix, you must input a certain number of pressure-flow specifications. The volume balance equations, resistance equations, and pressure-flow relation equations make up a large number of equations in the pressure-flow matrix. However, you should be aware of the specifications that are needed before the matrix will solve.

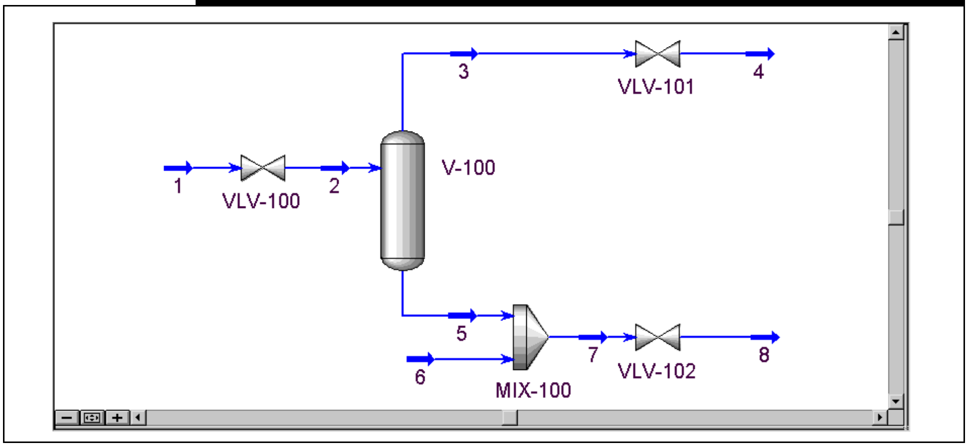
*For more information on specifying Pressure-Flow specifications for a material stream, see **Chapter 3 - Streams**.*

Degrees of Freedom Analysis

In almost all cases, a flowsheet being modelled dynamically using pressure-flow will require one pressure-flow specification per flowsheet boundary stream. A flowsheet boundary is one that crosses the model boundary and is attached to only one unit operation. Examples of such streams are the model's feed and product streams. All other specifications for the flowsheet will be handled when each unit operation is sized using the conductance or valve flow coefficient.

The following example confirms the "one P-F specification per flowsheet boundary stream" rule. In **Figure 1.14**, since there are 4 flowsheet boundary streams, you are required to make 4 pressure-flow specifications in order for the pressure flow matrix to solve. Note that the pressure flow specifications do not necessarily have to be set for each flowsheet boundary stream. Specifications can be made for internal flowsheet streams as long as there is one P-F specification per flowsheet boundary stream.

Figure 1.14



In the flowsheet shown above, there are 8 streams and 1 vessel holdup. In order to fully define the pressure flow matrix, the pressure and flow for each material stream and the pressure of each holdup must be solved for. In short, two variables are required for each material stream and 1 variable is required for each holdup:

$$\begin{array}{l} \textit{8 material streams} \times 2 \\ \underline{+\ \textit{1 vessel holdup} \quad \times 1} \\ \textit{17 pressure-flow variables} \end{array} \tag{1.23}$$

The accumulation or amount of holdup is solved using material balances in the holdup model. Although the holdup is not solved by the pressure-flow matrix, it is used by the volume balance equation to calculate the vessel pressure of the holdup which is a variable in the matrix.

The pressure and flow of material streams are named $P_{stream\ name}$ and $F_{stream\ name}$, respectively. The pressure of the holdup is named P_H. There are a number of equations which describe the relationship between the pressures and flows in this network. They are as follows:

Pressure-Flow Equation	Description	# of Eqns
Separator		
Volume Balance equations	$\frac{dP_H}{dt} = f(P, T, holdup, flows)$ The volume balance relates P_H with F_2, F_3 and F_5.	**1**
General Pressure relations	$P_H = P_2 = P_3 = P_5$ If the static head contribution in the integrator is not checked, this general pressure relation will be observed.	**3**
Valves		
Resistance equations	$F_2 = K_{VLV100}\sqrt{P_1 - P_2}$ $F_4 = K_{VLV101}\sqrt{P_3 - P_4}$ $F_8 = K_{VLV102}\sqrt{P_7 - P_8}$ This is the general form of the valve resistance equation. The actual equations vary according to inlet stream conditions.	**3**
General Flow relations	$F_1 = F_2$ $F_3 = F_4$ $F_7 = F_8$ Since the valves are usually not specified with holdup, this relation will be observed.	**3**

Pressure-Flow Equation	Description	# of Eqns
Mixer		
General Pressure relations	$P_5 = P_6 = P_7$ The equalize option is recommended for the operation of the mixer in dynamic mode. If this option is checked, this general pressure relation will be observed.	**2**
General Flow relation	$F_7 = F_5 + F_6$ Since the mixer is usually not specified with holdup, this relation will be observed.	**1**
Total Number of Pressure Flow Equations		**13**

With 17 variables to solve for in the network and 13 available equations, the degrees of freedom for this network is 4. Therefore, 4 variables need to be specified to define this system. This is the same number of flowsheet boundary streams.

Pressure-Flow Specification Guidelines

The previous section outlined the number of pressure-flow specifications that are required by the flowsheet in order for the degrees of freedom to be satisfied. This section presents possible PF specifications that can be made for the inlet and exit streams of *stand alone* operations. The purpose of this section is to demonstrate the range of specifications that can be made for different unit operations in HYSYS. It is hoped that this section will provide insight as to what should and should not be specified for each unit operation.

Valve

Rating information for the valve operation including the valve type and Cv values can be input on the **Sizing** page in the **Ratings** tab.

The dynamic valve can either be specified as having a set pressure drop or a pressure flow relation. This option is set on the **Specs** page of the **Dynamics** tab in the **VALVE** property view.

> Note that the P-F spec option for conductance-type unit operations should be used as much as possible since it is much more realistic in determining pressure flow relations in an actual plant. The pressure drop option is provided to ease the transition between **Steady State** and **Dynamic** mode. The pressure drop option may help more difficult simulations run since the initial exit stream conditions of the valve can be easily calculated using the pressure drop option.

1. For a pressure drop specification on the valve: one pressure spec and one flow spec is required for the inlet and exit streams.
2. For a pressure-flow specification on the valve: two pressures may be specified or one pressure and one flow

Pressure and level control can be achieved in a separator using valves on the vapour and liquid streams, respectively. It is best to use a pressure specification downstream of each valve. The percent openings on each valve can then be used to control the flow through each valve with a PID controller.

Heat Exchanger/Cooler/Heater

The heat exchange operations, like the valve, should use the P-F spec option as much as possible to simulate actual pressure flow relations in the plant.

The dynamic **HEAT EXCHANGER** can be specified as having a set pressure drop or a Overall K-Value (pressure-flow) relation. This option is set on the **Specs** page of the **Dynamics** tab in the **HEAT EXCHANGER** property view:

1. For a pressure drop specification on either the tube side or shell side: one pressure spec and one flow spec is recommended.
2. For a K-value spec on either the tube or shell side: two pressures may be specified or one pressure and one flow

K-values can be calculated using the ***Calculate K*** button on the **Specs** page of the **Dynamics** tab in the operation's property view.

HEATER and **COOLER** operations are much like **HEAT EXCHANGERS**. However, they only have a single K-value on their process side.

Separators

Rating information including the volume of the vessel, boot capacity, and nozzle location can be input on the **Sizing** and **Nozzles** pages in the **Ratings** tab.

A separator with no valves attached to the inlet and exit streams requires only one pressure specification. The other two streams are specified with flows. A more stable way to run the separator is to attach valves to the inlet and exit streams of the vessel. The boundary streams of the separator with valves should be specified with pressure.

Condenser/Reboiler

Rating information for the condenser and reboiler including the vessel volume, boot capacity, and nozzle location can be input on the **Sizing** and **Nozzles** pages of the vessel's **Ratings** tab.

It is highly recommended that the proper equipment be added to the reflux stream (e.g. pumps, valve, etc.). In all cases, level control for the condenser should be used to ensure a proper liquid level.

The **Partial Condenser** has three exit streams: the overhead vapour stream, the reflux stream, and the distillate stream. All three exit streams must be specified when attached to the main tray section. One pressure specification is recommended for the vapour stream. The other two exit streams must be specified with flow rates. Another option is to specify a **Reflux Flow/Total Liq Flow** value on the **Specs** page in the **Dynamics** tab. In this case, only one flow spec is required on either the reflux or distillate stream.

The **Fully-Refluxed Condenser** has two exit streams: the overhead vapour stream and the reflux stream. One pressure and flow specification is required for the two exit streams.

A **Fully-Condensed Condenser** has two exit streams: the reflux stream and the distillate stream. 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.

The **Reboiler** has two exit streams: the boilup vapour stream and the bottoms liquid stream. 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.

Separation Columns

For all separation columns, the tray section parameters including the tray diameter, weir length, weir height, and tray spacing can be specified on the **Sizing** page in the **Ratings** tab of the **Main TS** property view.

The basic **ABSORBER** column has two inlet and two exit streams. When used alone, the **ABSORBER** has four boundary streams and therefore requires four pressure-flow specifications. A pressure specification will always be 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.

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 will always be required for the liquid product stream leaving the bottom of the column.

The **REBOILED ABSORBER** column has a single inlet and two exit streams. When used alone, the **REBOILED ABSORBER** has three boundary streams and therefore requires three pressure-flow specifications; one pressure and two flow specifications. A pressure specification will always be required for the vapour product leaving the 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 8.2.2 - Condensers and Reboiler**.

Compressor/Expander/Pump

Rating information for the dynamic compressor, expander, and pump operations can be input on the **Curves** and **Inertia** pages in the **Ratings** tab.

In general, two specifications should be selected in the **Dynamics Specifications** group in the **Specs** page of the **Dynamics** tab in order for these unit operations 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 Increase** box should not be selected if the inlet and exit stream pressures are specified.
- The **Speed** box should not be selected if the **Use Characteristic Curves** check box is not selected.

The **COMPRESSOR**, **EXPANDER**, and **PUMP** operations have one inlet stream and one exit stream. Two pressures may be specified for the inlet and exit streams or one pressure and one flow may be specified.

Mixer / Tee

The dynamic **MIXER** and **TEE** operations are very similar. It is recommended that the **MIXER** be specified with the **Equalize All** option in **Dynamic** mode. It is also recommended that the dynamic **TEE** not use the dynamic splits as specifications. These options are set on the **Specs** page of the **Dynamics** tab in their respective operation views.

By specifying the dynamic **MIXER** and **TEE** as recommended, the pressure of the surrounding streams of the unit operation are equal if static head contributions are not considered. This is a realistic situation since the pressures of the streams entering and exiting a mixer or tee must be the same. With the recommended specifications, flow to and from the unit operations is by the pressure flow network and not by you, the user.

A number of streams can enter or exit a mixer or tee. For stand alone operations, one stream must be specified with pressure. The other inlet/exit streams are specified with flow.

1.4 Dynamic Operations: General Guidelines

This section outlines some guidelines or steps that you may follow in order to create and run a simulation case in **Dynamic** mode.

It is possible to create a case directly in **Dynamic** mode. Unit operations can be added just as easily in **Dynamic** mode as in **Steady State**. The integrator should be run after each addition of a unit operation in order to initialize exit stream conditions for the added unit operations.

It is also possible for you to build a dynamics case by first creating the case in **Steady State** mode. You can make the transition to **Dynamic** mode with some modifications to the flowsheet topology and stream specifications. **Section 1.4.2 - Moving from Steady State to Dynamics** outlines some general steps you can take in order to create a dynamics case from steady state mode. The Dynamic Assistant (**Section 2.1 - Dynamics Assistant**) can be used to quickly modify the steady state flowsheet so that it has a correct set of pressure flow specifications. It is important to note, however, that not all the modifications suggested by the Assistant will result in a stable pressure flow matrix for the PF solver.

It is suggested that you -when first learning dynamics- build simple cases in **Steady State** mode so that the transition to **Dynamic** mode is relatively easy. Once the transition from **Steady State** to **Dynamic** mode is made, other unit operations can easily be added to better model the process system. If you are more experienced, you may adopt different and more efficient ways to create a dynamics case.

1.4.1 Specification Differences between Dynamic and Steady State

It is apparent that the specifications required by the unit operations in **Dynamic** mode are not the same as the **Steady State** mode. This section outlines the main differences between the two modes in regards to specifying unit operations.

Steady State

The **Steady State** mode uses **modular** operations which are combined with a **non-sequential** algorithm. Information is processed as soon as it is supplied by you. The results of any calculation are automatically propagated throughout the flowsheet, both forwards and backwards.

Material, energy, and composition balances are considered at the same time. Pressure, flow, temperature, and composition specifications are considered equally. For instance, a column's overhead flow rate specification may be replaced by a composition specification in the condenser. The column may solve with either specification.

Dynamics

Material, energy, and composition balances in **Dynamic** mode are not considered at the same time. Material or pressure-flow balances are solved for at every time step. Energy and composition balances are defaulted to solve less frequently. Pressure and flow are calculated **simultaneously** in a pressure-flow matrix. Energy and composition balances are solved in a **modular sequential** fashion.

Because the pressure flow solver exclusively considers pressure-flow balances in the network, P-F specifications are separate from temperature and composition specifications. P-F specifications are input using the "one P-F specification per flowsheet boundary stream" rule. Temperature and composition specifications should be input on every boundary feed stream entering the flowsheet. Temperature and composition are then calculated sequentially for each downstream unit operation and material stream using the holdup model.

Unlike in **Steady State** mode, information is not processed immediately after being input by you. The integrator should be run after the addition of any unit operation to the flowsheet. Once the integrator is run, stream conditions for the exit streams of the added unit operation will be calculated.

1.4.2 Moving from Steady State to Dynamics

You should be aware that flow in the plant occurs as a result of the pressure-flow relations between nodes. Before a transition from **Steady State** to **Dynamic** mode occurs, the simulation flowsheet should be set up so that a definite pressure drop exists across the plant. The following outline indicates some basic steps you can take to set up a case in **Steady State** mode and then switch to **Dynamic** mode.

1. Adding Unit Operations

Identify material streams which are connected to two unit operations with no pressure flow relation and whose flow must be specified in **Dynamic** mode. These unit operations include the **SEPARATOR** operation and tray sections in a **COLUMN** operation. Add unit operations, such as **VALVES**, **HEAT EXCHANGERS**, or **PUMPS**, which define a pressure flow relation to these streams. It is also possible to specify a flow specification on this stream instead of using an operation to define the flow rate.

2. Equipment Sizing

Size all the unit operations in the simulation using actual plant equipment or predefined sizing techniques. Sizing of trays in columns can be accomplished using the Tray Sizing utility available from the **Utilities** page. Vessels should be sized to accommodate actual plant flowrates and pressures while maintaining acceptable residence times.

General Equipment Sizing Rules

Vessels (SEPARATORS, CONDENSERS, REBOILERS) should be sized for 5 - 15 minutes of liquid holdup time. Sizing and Costing calculations may also be performed using the Vessel Sizing utility in the **Sizing** page of the **Rating** tab.

Valves should be sized using typical flowrates. The valve should be sized with a 50% valve opening and a pressure drop between 15 and 30 kPa.

Column Tray Sizing Rules

Tray Sizing can be accomplished for separation columns using the Tray Sizing utility in the **Utilities** page. Any use of utilities should be restricted to **Steady State** mode. The trays are sized according to the existing flow rates and the desired residence times in the tray. Important variables include:

- Tray diameter
- Weir length
- Weir height
- Tray spacing

3. Adjusting Column Pressure

In steady state, the pressure profile of the column is user specified. In dynamics, it is calculated using dynamic hydraulic calculations. If the steady state pressure profile is very different from the calculated pressure drop, there may be large upsets in flow in the column when the integrator is run. A reasonable estimate of the column's pressure profile can be calculated using the Tray Sizing utility. This utility provides a *Max ΔP/Tray* value in the **Results** tab. The column pressure profile may be calculated using this value, the *Max ΔP/Tray* value, and a desired pressure specification anywhere on the column.

You may wish to change the *Max ΔP/Tray* value in order to achieve a desired pressure profile across the column. This can easily be done by modifying the *Weir height* in the **Ratings** tab in the Tray Sizing utility. Reducing the weir height lowers the static head contributions and lowers the *Max ΔP/Tray* value. In **Dynamic** mode, the *Nozzle Pressure Flow K-factors* (found on the **Dynamics** tab of the **Main TS** property view) may also be adjusted to better model the pressure drop across the column.

4. Logical Operations

Some logical operations from the steady state will be ignored. The **ADJUST** operation may be replaced by **PID Controllers**. The **RECYCLE** operation is redundant in **Dynamic** mode.

5. Adding Control Operations

Identify key control loops that exist within the plant. Implementing control schemes increases the realism and stability of the model. Disturbances in the plant can be modelled using the **Transfer Function** operation. The Events Scheduler can be used to model automated shutdowns and startups.

6. Enter the HYSYS Dynamic Environment

Dynamic Mode Button

Click on the ***Dynamic Mode*** button to switch from **Steady State** mode to **Dynamic** mode.

7. Adding Pressure-Flow Specifications

Specify one pressure-flow specification for each flowsheet boundary stream. For more information regarding pressure-flow specifications for individual unit operations, refer to **Section 1.3.3 - Pressure Flow Specifications**. Controllers play a large part in stabilizing the PF Solver. For more information regarding the implementation of controllers in HYSYS, see **Section 11.4.4 - Basic Steps in setting up a Control Strategy in HYSYS**.

Precautions

1. You should pay special attention to equipment with fixed pressure drops. Any fixed pressure drop specifications in equipment may yield unrealistic results, such as flow occurring in the direction of increasing pressure. Remember to check for fixed pressure drops in the reboiler and condenser of columns.
2. Be cautious of Heaters/Coolers with fixed duties. This may cause problems if the flow in the heater/cooler happens to fall to zero. It is recommended to use a controller, or a **Spreadsheet** function, or a temperature specification to control the temperature of a stream.
3. Feed and product streams entering and exiting tray sections should be at the same pressure as the tray section itself. Any large pressure differences between a feed or product stream and its corresponding tray section may result in large amounts of material moving into or out of the column.

It may be necessary to isolate and converge single pieces of equipment in the plant using the **Ignored** feature for each unit operation if there is an especially large number of unit operations in the flowsheet.

Run the **Integrator** after any unit operation has been added in **Dynamic** mode. Unlike the steady state environment, the exit streams of unit operations in **Dynamic** mode are not calculated until the Integrator is run. The Integrator should be run long enough to obtain reasonable values for the exit streams of the new operations.

8. Troubleshooting

Error messages may be displayed once the integrator is run:

Too many specifications/ Not enough specifications

The "*Too many specifications*" message indicates that HYSYS.Plant has detected too many specifications. The **Equation Summary View** window that appears with the message can provide help indicating the specification that is most likely not required. Press the ***Full Analysis*** button (or ***Partitioned Analysis*** button, if it is made available). At this point, HYSYS examines possible problem areas with the simulation case. Pressing the **Extra Specs** tab reveals the variable(s) most likely not required by HYSYS.

The "*Not enough specifications*" message indicates that the simulator has detected too few specifications. The **Extra Specs** tab in the **Equation Summary View** indicates possible variables that may be missing from the simulation case. The Dynamics Assistant can aid in identifying which P-F specifications should be added or deleted from a dynamic simulation case.

Singular Problem

This message indicates that not all of the equations in the PF solver matrix are independent of one another. This occurs when one or more equations are redundant. For instance, if a **VALVE** operation is using a pressure drop specification, the inlet and exit streams cannot both be specified with pressure. The pressure drop equation becomes redundant. It may be useful to overspecify a singular problem. HYSYS might be able to identify the redundant pressure flow specification and allow the case to solve.

The Pressure Flow Solver failed to converge

This message indicates that one or more pressure-flow specifications may be unreasonable. This message may also appear if there are sudden large upsets to the simulation case. It is helpful to enter the **Equation Summary View** to identify problem areas in the flowsheet. Press the ***Full Analysis*** button (or ***Partitioned Analysis*** button, if it is made available). By pressing the ***Update Sorted List*** button in the **Unconverged** tab, HYSYS reveals the type of equation, location, and scaled error associated with the unconverged nodes in the flowsheet.

Pay special attention to the unit operations with the largest errors in the **Uncoverged** tab. Check the vessel volumes of the uncoverged unit operations and ensure that they have been sized with reasonable residence times. Check the size of the valves attached to the unconverged unit operations.

For more information regarding the **Equations Summary View**, see Chapter 2 - Dynamic Tools.

1.5 Plant+

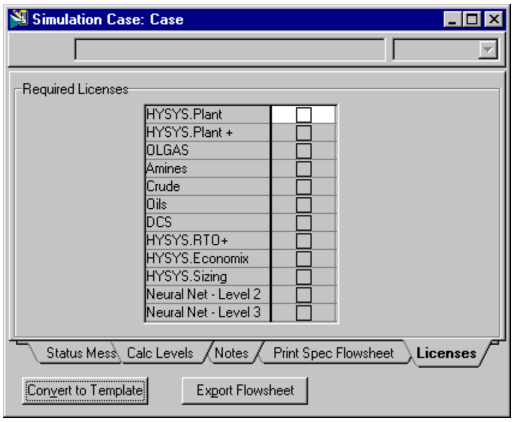
Plant+ is an extension of HYSYS that provides advanced dynamic features to your simulation. Plant+ allows you to put together very detailed models for operator training work or detailed dynamic studies. The capabilities exclusive to Plant + are as follows:

- Static head included in the pressure relationships. Users also have the ability to modify equipment elevations.
- Nozzle locations can be modified. For example an overhead vapour nozzle may be somewhere below the top of the vessel.
- Detailed valve actuator dynamics. The dynamics of the valve opening and closing are included in the model.
- A detailed heat loss model to take into account heat loss from vessels with holdup to the environment. For example, users may supply details about the equipment and insulation to take into account heat transfer from the vessel to the environment.
- Details on rotating equipment. Inertia terms will account for the starting up and shutdown of rotating equipment.

In order to use the Plant+ features you are required to purchase the Plant+ license. If you do not have a Plant+ license or it has not been activated you will not be able to see or access these features.

To activate your Plant+ features go to the simulation environment, open the **Simulation** menu and select **Main Properties**. Select the **Licenses** tab, located on the very right.

Figure 1.15



Click the check box that corresponds with HYSYS.Plant+ to activate the license. Note that once you have activated this license it can **NOT** be deactivated.

1.5.1 Compressible Gas Pipe

The Compressible Gas Pipe Model uses an algorithm that solves a vector system using the Two-Step Lax-Wendroff method with Boris & Book anti-diffusion.

The Compressible Gas Pipe 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 a either boundary pressure.

Compressible Gas Pipe Button

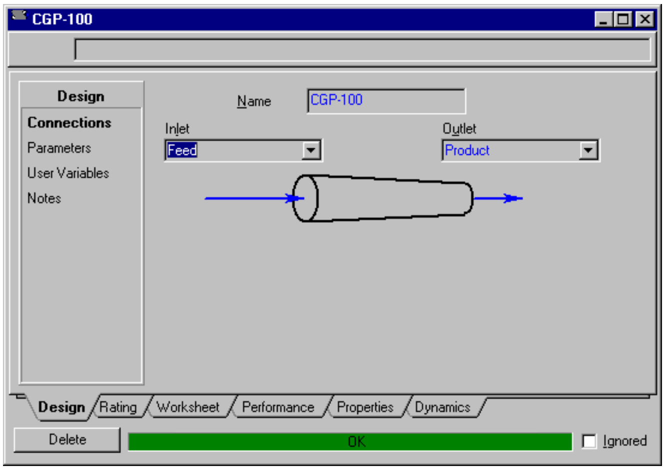
To install the **Compressible Gas Pipe** operation, press **F12** and choose **Piping Equipment** from the **UnitOps** view or click the Compressible Gas Pipe button in the Object Palette.

1.5.2 Design Tab

The **Design** tab provides access to four pages: **Connections**, **Parameters**, **User Variables** and **Notes** page.

Connections Page

Figure 1.16

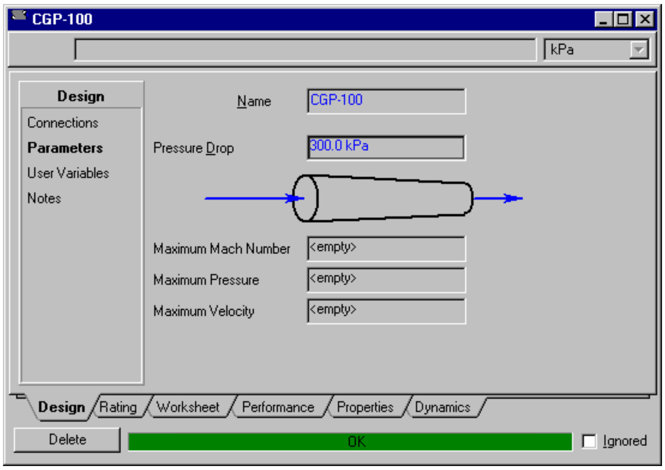


On the **Connections** page, you must specify the **Inlet** and **Outlet** material streams. This can be done by selecting existing streams from the drop down lists associated with the Inlet and Outlet fields. You can also create new streams by selecting the particular field and entering the new stream name in the Edit Bar. You may also edit the operation's name on this page.

Note that the Compressible Gas Pipe does not support an energy stream.

Parameters Page

Figure 1.17



The **Parameters** page simply allows you to specify the Pressure Drop across the pipe as well as the operation's name. There are also three calculated values that are displayed on the page.

- Max. Mach Number - For steady state calculations this will always be 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 will always be 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 will always be at the outflow from the pipe. During dynamic calculations this can be at any location within the pipe

User Variables Page

The **User Variables** page allows the user to create and implement variables in the HYSYS simulation case. For more information on implementing the User Variables option, see the **User Variables** chapter in the **Customization Guide**.

Notes Page

The **Notes** page provides a text editor where you can record any comments or information regarding the **Compressible Gas Pipe** or pertaining to your simulation, in general.

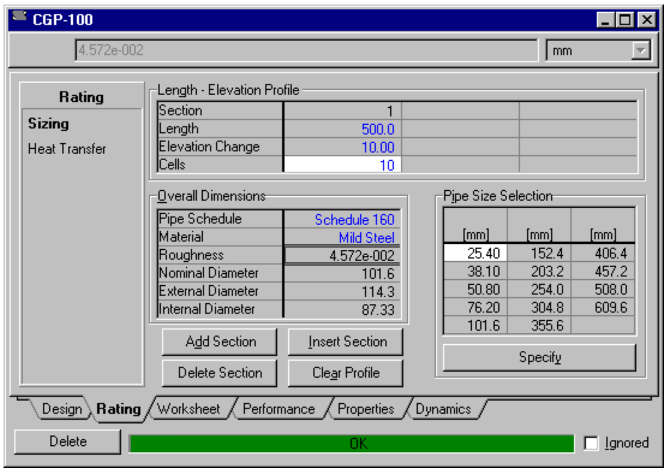
1.5.3 Rating Tab

The **Ratings** tab provides access to two pages: Sizing and Heat Transfer. On the **Sizing** page you provide 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 **Compressible Gas Pipe** is constructed. You can provide details for each fitting or pipe section that is contained in the **Compressible Gas Pipe** that you are modeling. An unlimited number of pipe sections or fittings can be added on this page.

Figure 1.18



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.

- Add Section button - Adds the new selection after the currently selected section.
- Insert Section button - Adds the new selection before the currently selected section

For each segment that you add, you must supply the following:

Field	Description
Length	The physical length of the pipe. Note 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 will 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:

$$\frac{\text{Cell Length}}{\text{Time Step}} < 0.5 \text{ Sonic Velocity} \quad (1.24)$$

To delete a section, click on the section you wish 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.

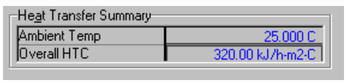
The Overall Dimensions group manages the pipe diameter and material data. This works in the same fashion as the standard Pipe Segment unit operation (refer to **Section 4.2 - Pipe Segment** in the Steady State Modelling book).

Note that the External Diameter is not currently used by the calculations. It has been added so that the heat transfer models may 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 1.19



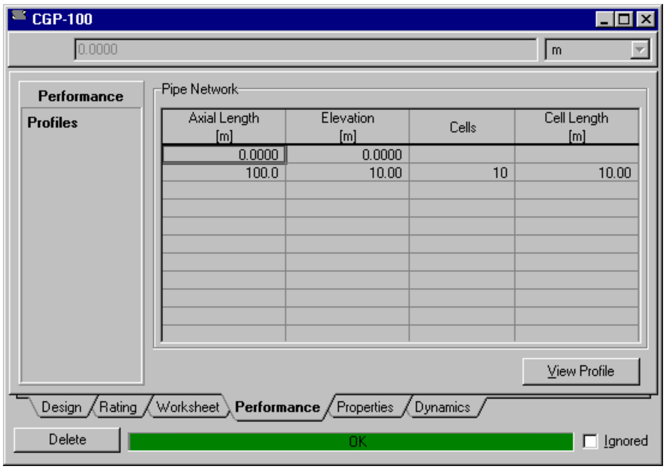
Where Ambient Temperature is the bulk ambient temperature and Overall HTC is the overall heat transfer coefficient based upon the inside diameter of the pipe.

1.5.4 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, Properties,** and **Composition** pages contain selected information from the corresponding pages of the **Worksheet** tab for the stream property view.

1.5.5 Performance Tab

Figure 1.20



This tab is functionally similar to the Performance tab on the standard Pipe segment unit operation (refer to **Section 4.2 - Pipe Segment**). It differs only on the properties displayed on the profile view. The following properties are visible:

- Axial Length
- Pressure
- Temperature
- Mass Flow
- Velocity
- Mach Number
- Mass Density
- Internal Energy
- Enthalpy
- Speed Of Sound

1.5.6 Properties Tab

Due to the number 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

$$H = C_p \Delta T \tag{1.25}$$

$$\rho = \frac{PMW}{RT} \tag{1.26}$$

Compressible Gas

As for perfect gas, but

$$\rho = \frac{PMW}{ZRT} \tag{1.27}$$

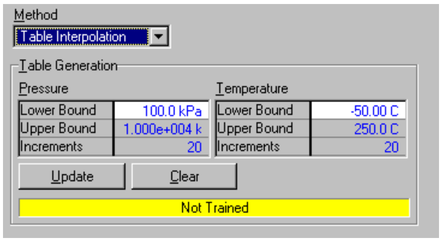
The compressibility factor, Z is calculated from the current property package for the flow sheet at the average conditions within the pipe.

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, the user must train the neural network.

Figure 1.21

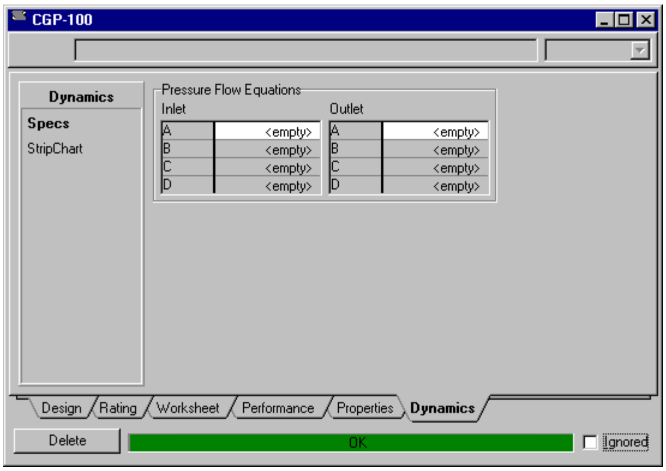


The Table Generation group manages the extent of the training. Care must be taken to train over the full extent of the expected range of operating conditions since extrapolation will always yield unpredictable results

1.5.7 Dynamics Tab

Specs Page

Figure 1.22



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 flow sheet equations. This is achieved by making perfurations at each end of the pipe for each time step and re-evaluating the change in state over the time step. These changes are the fit to an equation of the following form which is passed to the Pressure Flow solver:

$$A.\mathrm{Pres} + BFlow^2 + CFlow + D = 0 \tag{1.28}$$

The Pressure Flow Equations group displays the values for the coefficients in the above equation, which will be continuously updated at each time step.

StripChart Page

The **Stripchart** page allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the **Create Stripchart** button to display the stripchart.

1.5.8 A Model For Single Phase Compressible Flows

The following equations are used in HYSYS to model a single phase compressible flow.

Governing Equations

Mass:

$$\frac{\partial(A\rho)}{\partial t} + \frac{\partial(A\rho u)}{\partial x} = 0 \tag{1.29}$$

Momentum:

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2 + p)}{\partial x} = \rho g \sin\theta - \frac{1}{2} f \rho u |u| \frac{S}{A} - \rho u^2 \frac{1}{A}\frac{dA}{dx} \tag{1.30}$$

Energy:

$$\frac{\partial(\rho E)}{\partial t} + \frac{\partial(\rho H u)}{\partial x} = k(T_{wall} - T)\frac{S}{A} - \rho g \sin\theta - \frac{1}{2} f \rho u^2 |u| \frac{S}{A} - \rho H u \frac{1}{A}\frac{dA}{dx} \qquad (1.31)$$

where: $A = \frac{1}{4}\pi D^2$

$E = e + \frac{1}{2}u^2$

$H = h + \frac{1}{2}u^2$

$S = \pi D$

and A - pipe cross-sectional area

D - pipe diameter

e - internal energy

E - total internal energy

f - friction factor

g - acceleration due to gravity

h - enthalpy

H - total enthalpy

k - heat transfer coefficient

p - pressure

S - pipe perimeter

t - time

T - temperature

T_{wall} *- wall temperature*

u - velocity

x - distance

θ *- pipe inclination*

ρ *- density*

Algorithm

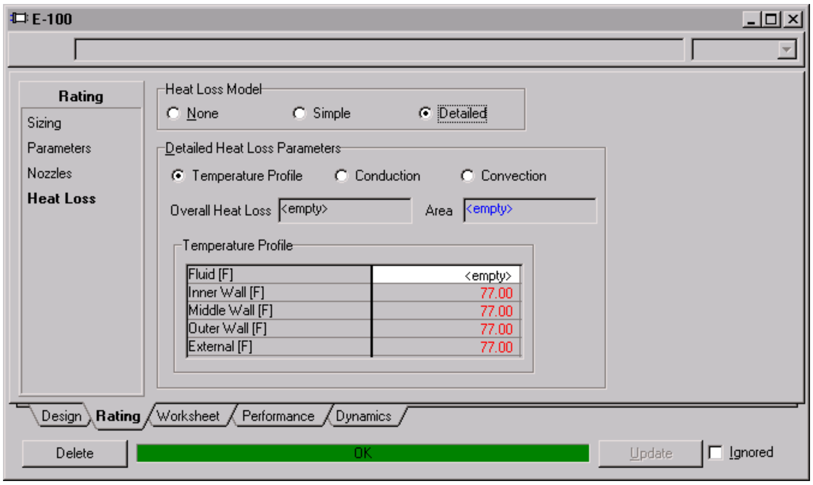
$$\frac{\partial U}{\partial t} + \frac{\partial D}{\partial x} = \underline{G} \tag{1.32}$$

The algorithm solves the vector system by the Two-Step Lax-Wendroff method with Boris & Book anti-diffusion.

1.5.9 Detailed Heat Model

The **Detailed Heat** model is located on the **Heat Loss** page of the **Rating** tab.

Figure 1.23



There are two values that are common to each of the three radio buttons found in the Detailed Heat Loss Model; the **Overall Heat Loss** value and the **Overall Heat Transfer Area**.

The other parameters that are displayed by selecting one of the three radio buttons are described in the table below.

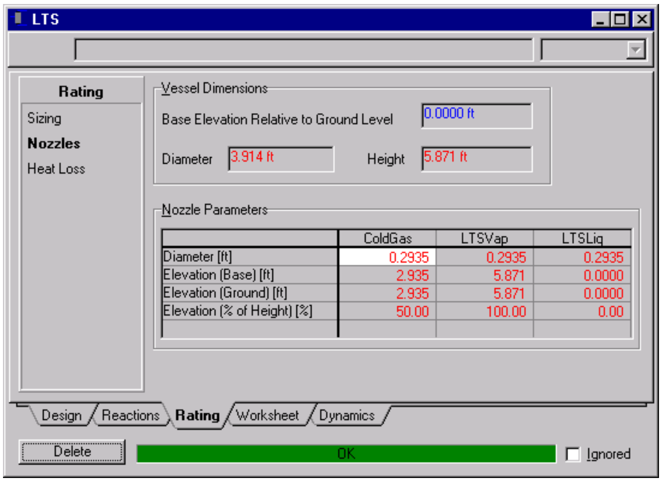
Radio Button	Description
Temperature Profile	Displays the temperatures of the various fluids, walls, insulation and surroundings. Note the parameters that appear on this page will vary between different unit operations.
Conduction	Displays the conductive properties of the **wall** and **insulation**. The following properties can be specified by you: • Conductivity of material • Thickness of material • Heat capacity of material • Density of material **Equation (1.14)** and **(1.15)** demonstrate how the parameters are used by the heat loss model. The heat transfer area is calculated from the vessel geometry. The rest of the heat transfer parameters may be modified. The insulation thickness can be specified as zero to model vessels without insulation. The metal wall must be specified with a finite thickness.
Convection	Displays the convective heat transfer coefficients for heat transfer within the holdup and heat transfer occurring from the outside the holdup to the surroundings. Both the inside and outside heat transfer coefficients may be modified from their default values.

The governing equations relating heat loss from the vessel and the **Detailed** heat loss parameters shown here are discussed in **Section 1.2.4 - Heat Loss Model**.

1.5.10 Nozzles

The **Nozzles** page, located on the **Rating** tab, contains information regarding the elevation and diameter of the nozzles.

Figure 1.24



The elevations of each nozzle attached to the piece of equipment are displayed relative to several reference points:

- The **Ground** is a common reference point from which all equipment elevations can be measured.
- The **Base** is defined as the bottom of the piece of equipment.

The following is a summary of the Nozzle Parameters section:

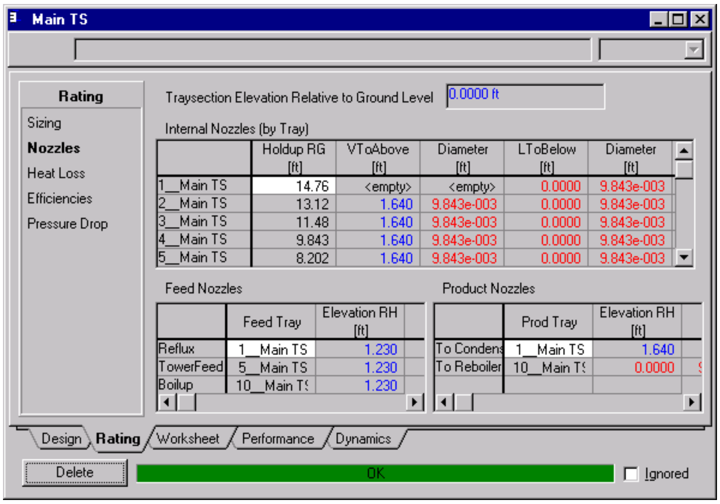
Nozzle Parameter	Description
Diameter	The nozzle diameter is displayed and can be modified in this cell. The nozzle diameter has physical meaning in relation to the liquid level in the holdup. If the liquid level sits across the nozzle, the phase fraction depends on how far the level is in relation to nozzle diameter.
Elevation (Base)	This is the elevation of the nozzle above the base (the bottom of the piece of equipment)
Elevation (Ground)	This is the elevation of the nozzle above the ground.
Elevation (% of Height)	This is the elevation of the nozzle as a percentage of the height of the vessel.

Nozzle Parameter	Description
Base Elevation relative to Ground	This is how far the piece of equipment is above the ground.
Diameter	Allows you to specify the Diameter of the vessel.
Height	Allows you to specify the Height of the vessel.

Main Tray Section

The **Nozzle** page for the Main Tray Section in the column environment is setup different from all the other unit operations.

Figure 1.25



The information is broken down into three groups: tray by tray for internal nozzles, Feed nozzles and Product nozzles. The following information is available within these groups.

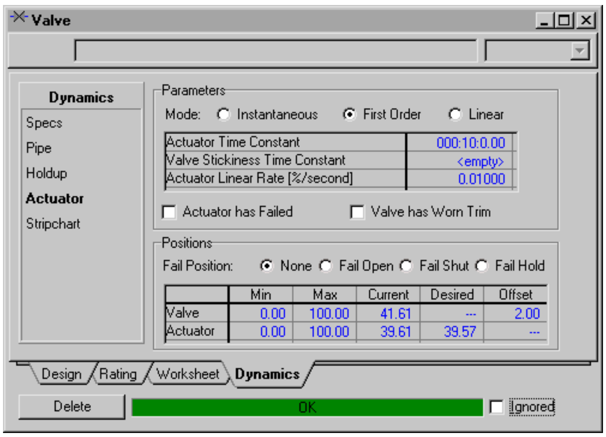
Object	Description
Traysection Elevation Relative to Ground	In this cell, you can specify the height of the tray section above the ground. The height is measured as the distance between the ground to the bottom tray.
Holdup RG	This column gives the height of each tray relative to the ground. The values are dependent on the tray spacing and the value entered in the **Traysection Elevation Relative to Ground** cell.

Object	Description
VToAbove	This column gives the elevation for vapour leaving the tray relative to the tray. By default this value is the tray spacing.
LToBelow	This column gives the elevation for liquid leaving the tray relative to the tray. By default this value is zero.
Elevation RH	This value can be specified for both the feed and product nozzles. This gives the elevation of the nozzles relative to the height of the column.
Diameter	Specifies the diameter of the nozzle for the Feed nozzles, Product nozzles, VToAbove and LtoBelow sections.

1.5.11 Control Valve Actuator

The **Actuator** page, located on the **Dynamics** tab of the **Valve** unit operation, allows you to model valve dynamics in the valve operation. This page also contains information regarding the dynamic parameters of the valve and the % open positions of the actuator and the valve.

Figure 1.26



A control valve in HYSYS consists of a valve and an actuator. They are defined as follows:

Valve component	Description
Actuator	An actuator is a device which applies the force required to cause movement in the valve.
Valve	The valve opening has a direct impact on the flow through the valve. This relationship is a function of the valve type and the pressure of the surrounding pieces of equipment.

In reality, changes that occur in the actuator are not observed instantaneously in the valve. Moreover, changes in the output signal of a controller, OP, do not instantaneously translate to changes in the actuator. Because the actuator and valve are physical items, they take time to move to their respective desired positions. This causes dynamic behaviour in actual control valves.

Valve Mode

The valve mode defines the relationship between the desired actuator position and current actuator position. The desired actuator position can be set by a **PID Controller** or **SPREADSHEET** operation. A controller's output, OP, for instance, is exported to the desired actuator position. Depending on the valve mode, the current actuator position can behave in one of three ways:

Instantaneous Mode

In this mode, the actuator moves instantaneously to the desired actuator position defined by the controller. The equation defining the relationship is:

$$Act\,\% = Act_{\text{Desired}}\,\% \tag{1.33}$$

First Order Mode

A first order lag can be modelled in the response of the actuator position to changes in the desired actuator position. The movement of the actuator is defined by the solution of the following differential equation:

$$\tau\frac{d(Act\%)}{dt} + Act\% = Act_{Desired}\% \tag{1.34}$$

The actuator time constant, τ, in **Equation (1.34)** can be specified in the **Actuator Time Constant** cell.

Linear Mode

The actuator can be modelled to move to the desired actuator position at a constant rate. The actuator moves according to the following equation (if the desired actuator position is above the current actuator position):

$$\begin{aligned} &Act\% = (\text{Actuator Linear Rate})\Delta t + Act_o\% \\ &\text{until } Act\% = Act_{Desired}\% \end{aligned} \tag{1.35}$$

The linear rate can be specified in the **Actuator Linear Rate** cell. Typical stroke times (closure rates) are as follows:

1. **Electric-Hydraulic Actuators**: approximately 12 inches/minute
2. **Piston Actuators** (Motor Driven): under 70 inches/minute

Valve Stickiness

In reality, the valve does not respond instantaneously to changes in the actuator. A first order lag can be modelled in the response of the actual valve position to changes in the actuator position. The behaviour of the valve percent opening as a function of the actuator position is shown as follows:

$$\tau_{sticky}\frac{d(Valve\%)}{dt} + Valve\% = Act\% + \text{Offset} \tag{1.36}$$

The valve stickiness time constant is specified in the **Valve Stickiness Time Constant** cell. The offset can be specified in the **Valve Position** section. If the valve stickiness time constant is left empty, the time constant value is assumed to be zero.

If the **Valve has Worn Trim** check box has been selected, a 0.1% offset is added to the right hand side of **Equation (1.36)**. This offset disallows the valve percent opening to fully close.

Fail Modes

Actuators usually have a fail-safe function. If there is a disruption to the power source driving the valve, the actuator will place the valve in a safe position, either fail open or fail close. Fail modes can be specified by pressing the corresponding radio button in the **Valve Position** section. The valve can be modelled to fail by activating the **Actuator has failed** check box.

Fail Open

In the event that the signal from the controller is cut off from the valve, the valve will become wide open. In HYSYS, if the **Fail Open** radio button in selected, the signal received by the valve is modified by the valve as follows:

$$Act_{\text{Desired}}\ \% = 100\% - Act_{\text{Desired}}\ \%\ (from\ controller) \qquad (1.37)$$

Note that if $\text{Act}_{\text{Desired}}\%$ (from controller) becomes zero in the event of a signal failure, the actuator becomes fully open. The fact that the signal from the controller is modified by the valve operation using **Equation (1.37)** has implications on the direction of the controller. If the **Fail Open** mode is chosen for the valve, reverse-acting controllers need to be toggled as direct-acting and direct-acting controllers need to be toggled as reverse-acting.

Fail Shut

In the event that the signal from the controller is cut off from the valve, the valve becomes fully closed. HYSYS does not modify the signal from the controller as with the **Fail Open** mode. If the signal from the controller becomes zero, so will the $\text{Act}_{\text{Desired}}\%$ value. Since the signal from the controller is not modified by the valve, the controller's direction does not have to be changed.

Fail Hold

In the event that the signal from the controller is cut off from the valve, the valve fails in the position it is currently in.

Valve Position

Various valve position parameters for the actuator and the valve are displayed in the **Valve Position** section:

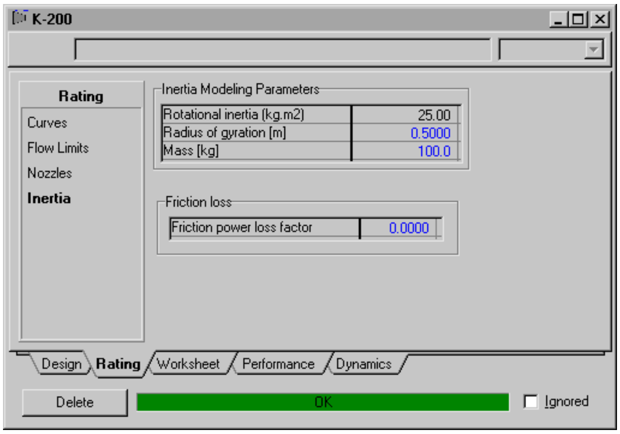
Valve Position Parameter	Definition
Minimum	The minimum position the actuator or valve can physically achieve. Leaky valves can be modelled by specifying a non-zero value for the minimum valve position.
Maximum	The maximum position the actuator or valve can physically achieve.

Valve Position Parameter	Definition
Current	The actual position of the actuator or valve in time.
Desired	The desired actuator position set by a **PID Controller** operation or imported from a **Spreadsheet** operation.
Offset	The Offset defined in **Equation (1.36)** can be specified in this cell.

1.5.12 Inertia

The inertia modelling parameters and the frictional loss associated with the impeller in the **Pump, Compressor** and **Expander** can be specified on the **Inertia** page of the **Rating** tab for these unit operations.

Figure 1.27



Not all of the energy supplied to a dynamic pump or compressor is transferred to the fluid. Likewise, not all the energy provided by an expander translates to kinetic energy. There are frictional losses associated with the moving parts of these unit operations. In a pump or compressor, power is also required to accelerate the rotating impeller and shaft. In general, the total power or duty supplied to or provided from a pump, compressor or expander has three parts:

- Rate of energy imparted to or provided by the fluid.
- Rate of energy required to accelerate the rotational speed of the shaft.
- Rate of energy lost due to mechanical friction loss.

The rate of energy supplied to the fluid can be observed in the enthalpy change between the inlet and exit streams. For a pump or compressor, this is:

$$\textit{Rate of energy imparted to the fluid} = F_1(MW)(h_2 - h_1) \tag{1.38}$$

where: h_2 *= the enthalpy of the exit stream*

h_1 *= the enthalpy of the inlet stream*

MW *= the molecular weight*

Inertial Modelling Parameters

The rate of energy required to accelerate the speed of a pump or compressor is a function of the rotational inertia of the impeller and the rotational speed. The rotational inertia, I, is calculated as follows:

$$I = MR^2 \tag{1.39}$$

where: M *= the mass of the impeller and rotating shaft*

R *= the radius of gyration*

The mass and radius of gyration can be specified in the **Inertial Modelling Parameters** section. The power required to accelerate the impeller, E_I, can be calculated using:

$$E_I = I|\omega|\frac{d\omega}{dt} \tag{1.40}$$

where: ω *= the rotation speed*

Friction Loss

The rate of energy lost from mechanical inefficiencies depends on the frictional power loss factor, f_{fric}, which can be specified in the **Friction Loss** section. The frictional work, E_f, can be calculated as follows:

$$E_f = f_{fric}I\omega|\omega| \tag{1.41}$$

A typical value for f_{fric} is 0.0001.

1.5.13 Static Head

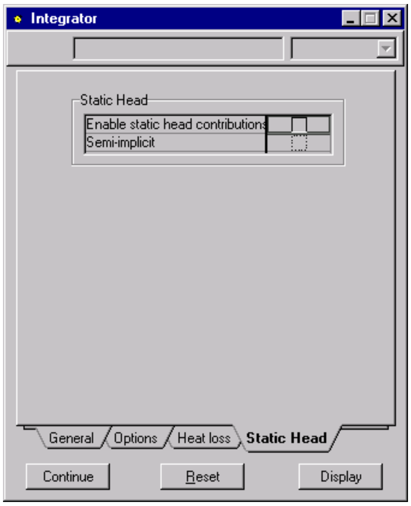
Note for the column operation the contribution of the internal levels is always calculated by HYSYS.

For any unit operations with holdup, HYSYS calculates the static head considering the equipment holdup, the geometry and the elevation of any attached nozzles. The **Static Head** page, found on the **Dynamics** tab of the **Main Tray Section** in the column environment, allows you to choose the calculation method used to calculate the static head for this operation. There are four options given on this page:

- Use global option in integrator (full on or off)
- No contributions for this operation
- Internal levels contribution only (partial)
- Levels and feed nozzle elevation differences (full)

In order for HYSYS to calculate the static head for any unit operation you need to enable the calculations. This is done on the **Static Head** page of the Integrator.

Figure 1.28



There are two check boxes available on this page:

Object	Description
Enable static head contributions	Activating this option tells HYSYS to include the effects of static head in the calculations.

1.5.14 Startup

The **Startup** page, on the **Rating** tab of the Pump unit operation, allows you to specify the **Typical operating capacity** in the **Design Flow** group. This parameter is used to aid in starting pumps up, which may have vapour in the line (e.g. due to a reverse flow). The existence of vapour can cause difficulty when the pump starts up. Hence, if the flow is less than a certain fraction of this typical operating capacity, then the density is compensated to help start the pump up. The **Typical operating capacity** value enables HYSYS to decides when it is reasonable to compensate the density.

2 Dynamic Tools

Modelling a process in dynamics is a complex endeavour. From the perspective of defining the model, you must consider parameters such as vessel holdups, valve sizing, and use of pressure flow specifications. To help simplify this process, HYSYS has several dynamic tools.

The first of these tools is the Dynamics Assistant. The Assistant provides a tool for easily converting old HYSYS dynamic cases to pressure flow dynamics. It provides general assistance to users who are learning how to create dynamic cases. It prepares steady state cases for dynamic simulation by ensuring that all the correct information is specified, thus avoiding over or under specified or singular problems.

The View Equations tool provides another means of analyzing cases for dynamic simulation. This tool provides a summary of the equations and variables used by the simulation when running in dynamics. By analyzing the case, it is possible to determine if there are required or redundant pressure flow specifications. In some instances, cases which are running in dynamics will fail to converge, in this case, the View Equations tool can be used to help determine what part of the simulation is causing problems.

The Integrator allows you to control some of the integration parameters which are used by HYSYS. Simple parameters such as the time step or the integration stop time or advanced parameters such as the execution rates of the different balances can be set from this tool. Once a case is running in dynamics, the current simulation time and the real time factor may be viewed.

For more advanced modelling, HYSYS has the Event Scheduler. Using this tool, HYSYS can perform predetermined actions at given times in the simulation; warn you by playing a sound when the temperature of a stream reaches a certain point, stop the integration once a condenser level stabilizes, or increase a feed rate after the simulation has run for a given time period.

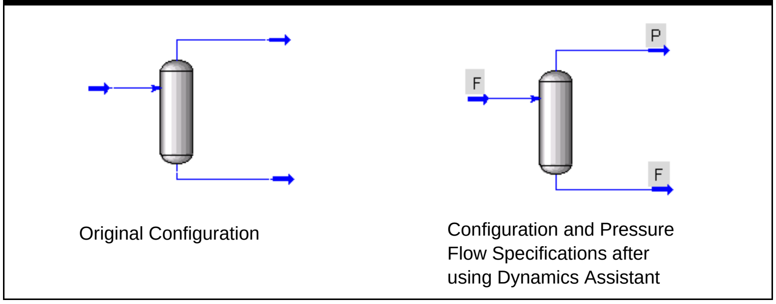
2.1 Dynamics Assistant

The Dynamics Assistant provides a quick method for ensuring that a correct set of pressure flow specifications is used. The Assistant can be used when initially preparing your case for dynamics, or when reading in an old HYSYS 1.x dynamic case.

> The Assistant makes **recommendations** for specifying your model in **Dynamic** mode. You do not have to follow all the suggestions. It is recommended that you are aware of the effects of each change you make.

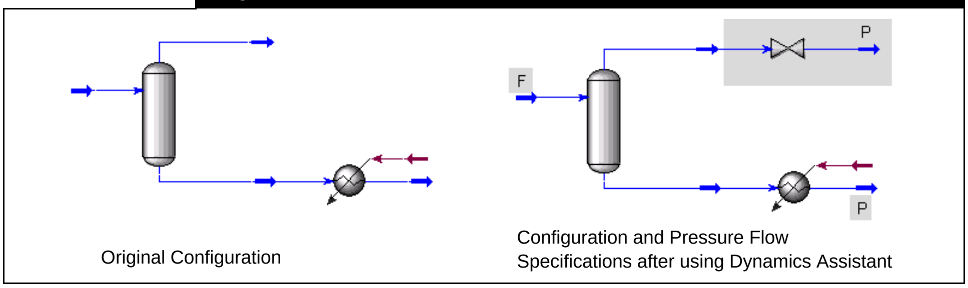
The Assistant recommends a set of specifications which is reasonable and guarantees that the case is not over or underspecified, or singular. It has an option of doing a quick examination for potential problems that can occur while moving from steady state to dynamics s as well as before running the case in dynamics. In the case of a simple separator, HYSYS will add pressure flow specifications as shown in the figure below.

Figure 2.1



However, in more complicated models such as the one shown in **Figure 2.2**, the Dynamics Assistant may recommend the insertion of valves in some terminal streams.

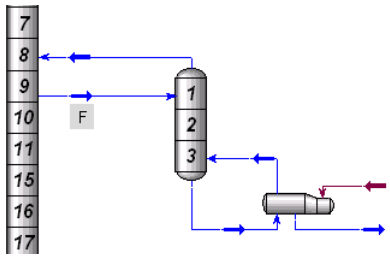
Figure 2.2



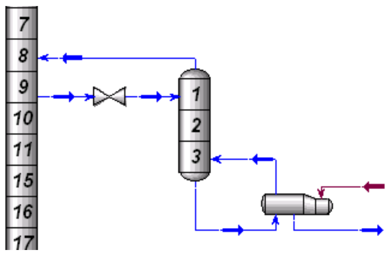
A Flow Specification warning from the Dynamics Assistant usually indicates that your model is missing some equipment.

Although the Pressure Flow specifications added by the Assistant are adequate for starting a case in Dynamics, detailed dynamic modelling may require more advanced modifications. In cases where unit operations such as separators are directly connected via multiple streams, the flow cannot always be determined. As a temporary fix the Assistant may add a flow specification. However, the user should add the missing unit operations (e.g. pumps, valves, etc.) to define the pressure-flow relation between the vessel unit operations.

Figure 2.3



Configuration of two units without Pressure Flow relationships when using the Dynamics Assistant.



Configuration of two units without Pressure Flow relationships when preparing for detailed Dynamic modelling.

In addition to ensuring that the correct Pressure Flow specifications are used for your dynamic case, the Assistant will size all necessary equipment that has not yet been sized. The parameters sized are: vessel volumes, valve Cvs and k values for equipment such as heaters, coolers, and heat exchangers. The assistant sizes required unit operations based on the flow conditions and specified residence times. The assistant also checks the Tray Section pressure profile for both steady state and dynamics model to ensure a smooth dynamics start. It also ensures that the tray section and attached stream have the same pressure.

As a final note, although the assistant ensures that your case will run in Dynamics, it is not intended that the changes made will be sufficient for your case to line out. It is still your responsibility, as the user, to ensure that an adequate control scheme is added to the case and that your model has been properly rated (i.e. existing vessels have been adequately sized).

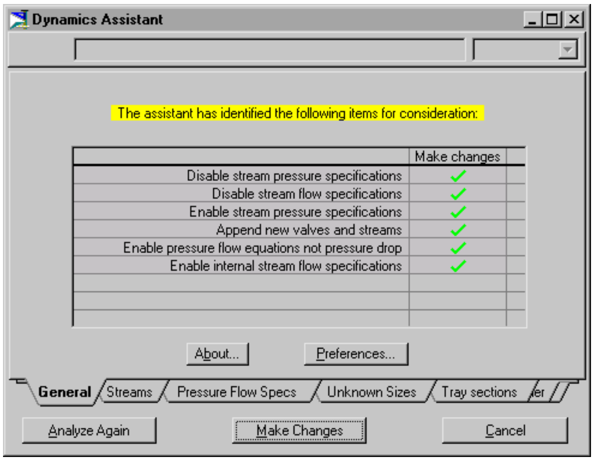
*The **Dynamics Assistant** button is only available on the **Equation Summary** view after the **Full Analysis** button has been pressed and if there are problems with the case.*

The Dynamics Assistant can be opened via three different methods.

- Using the hot key combination of **CTRL Y**
- Selecting **Dyn Assistant** from the **Tools** Menu Bar
- Pressing the ***Dynamics Assistant*** button from the Equation Summary View.

2.1.1 General Tab

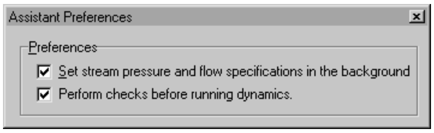
Figure 2.4



The **General** tab contains a summary of the changes which HYSYS recommends for dynamic simulation. Each item in the list has either a green check mark or a red 'x' located to the right of the item indicating whether the change will be made. The check mark indicates that the change will be made while the 'x' indicates that the change will not be made. You can toggle between the two states by clicking on the check or 'x' with the primary mouse button. Double clicking on any entry in the list will open the page associated with the change.

The ***Preferences*** button opens the Assistant Preferences view (see Figure 2.5; which allows you to change the way Dynamics Assistant will run. The **Set stream pressure and flow specifications in the background** check-box allows the Assistant to check and uncheck stream pressure and flow specifications as it sees fit. If the **Perform checks before running dynamics** check box is enable the Dynamics Assistant checks for any missing specification which can cause potential problems in dynamic simulation before you switch to **Dynamic** mode or run the **Integrator**.

Figure 2.5



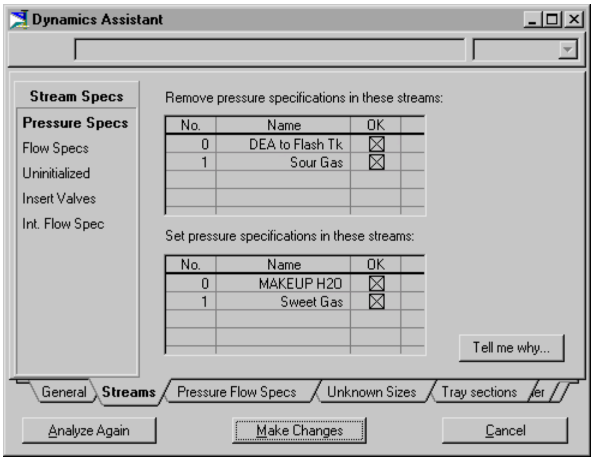
The ***Analyze Again*** button causes HYSYS to re-evaluate the simulation case. This is particularly useful if, while the Dynamics Assistant is open, you make changes to the case, which affect the setup of the simulation for dynamics.

The ***Make Changes*** button causes HYSYS to start making all the enabled changes within the Assistant.

2.1.2 Streams Tab

Pressure Specs Page

Figure 2.6



The **Pressure Specs** page, lists all the streams which will have pressure specifications either added or removed. The list of streams in the **Remove pressure specifications in these streams** group corresponds to the streams which currently have pressure specifications that the Assistant suggests you remove. Streams contained in the **Set pressure specifications in these streams** group are those where a pressure specification is recommended.

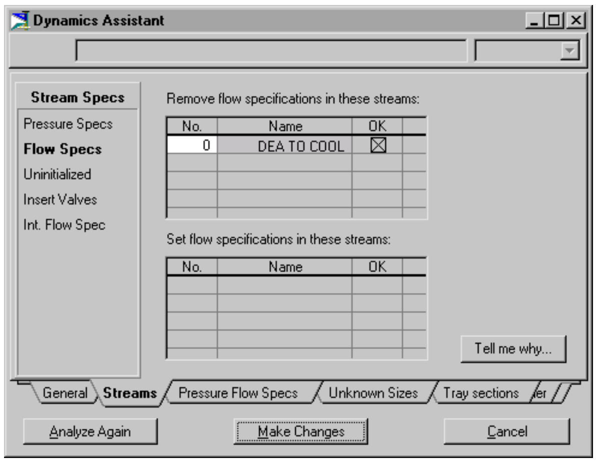
As a rule, pressure specs should be added to boundary streams and not internal streams whose pressure should be determined by the surrounding equipment. If the Assistant plans to insert a valve, the stream in question will be shown as not needing a pressure specification because the new stream at the other end of the valve will receive the pressure specification.

*If you do not understand the reasoning behind the Dynamics Assistant's recommendations select the **Tell me why** button for a brief explanation.*

If you do not want HYSYS to change the pressure specification for a stream listed in either group, uncheck the **OK** box for the given stream. This will prevent HYSYS from making the pressure specification change for the stream. To view a stream in either list, double click on the stream to open its property view.

Flow Specs Page

Figure 2.7



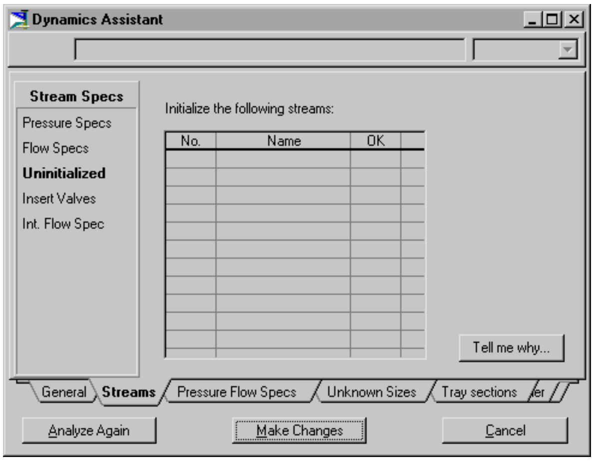
The **Flow Specs** page, lists all the streams which will have flow specifications either added or removed. The list of streams in the **Remove flow specifications in these streams** group corresponds to the streams which currently have flow specifications which are not recommended, and will be removed. Streams contained in the **Set flow specifications in these streams** group are those which require a flow specification as a temporary measure.

Flowrates should be determined by pressure differences or equipment such as pumps and compressors. For simplicity, the Dynamics Assistant may add flow specifications to feed streams instead of adding additional equipment.

If you do not want HYSYS to change the flow specification for a stream listed in either group, uncheck the **OK** box for the given stream. This will prevent HYSYS from making the flow specification change for the stream. To view a stream in either list, double click on the stream to open its property view.

Uninitialized Page

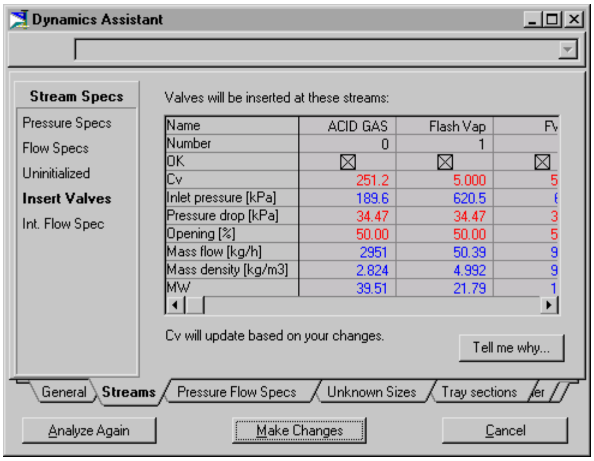
Figure 2.8



The **Uninitialized** page contains the list of streams which are not completely defined. For HYSYS to initialize any streams listed on this page, you must be in **Dynamic** mode; uninitialized streams will not be initialized when in **Steady State** mode. Cases saved in HYSYS 1.2 dynamics lack stream phase information. This information can be replaced by flashing the streams in question. Switching back to steady state, solving, then returning to **Dynamic** mode will also fix these streams. Streams with no values will receive initial estimates for temperature and pressure of 25°C and 101.33 kPa respectively.

Insert Valves Page

Figure 2.9

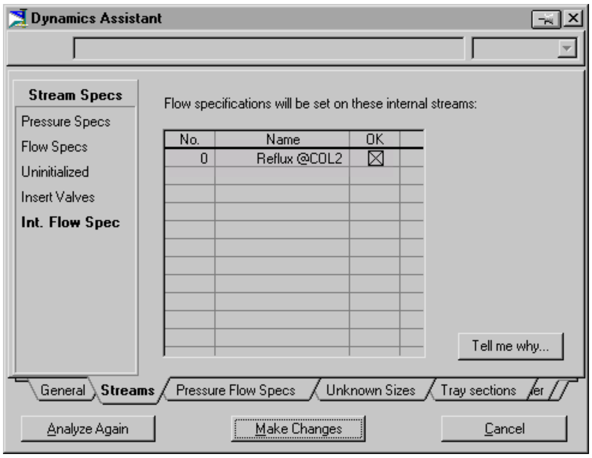


The **Insert Valves** page lists the valves which HYSYS will insert to ensure the pressure flow specifications are not singular (i.e. the pressure-flow matrix is unsolvable). HYSYS will attempt to attach valves to boundary streams that are connected to unit operations without pressure flow (flow proportional to pressure difference) capability. The outlet stream for the valve will automatically be assigned a pressure specification. Both the outlet stream and the valve added by HYSYS are named according to the original boundary stream; a valve attached to boundary stream 4 will be named VLV-4 and the outlet stream 4-1.

A situation may arise where the Dynamics Assistant will recommend the addition of a valve on a stream where you do not wish to add one. In this case, you should ensure that those streams receive either a *flow* or *pressure specification*.

Internal Flow Specs Page

Figure 2.10

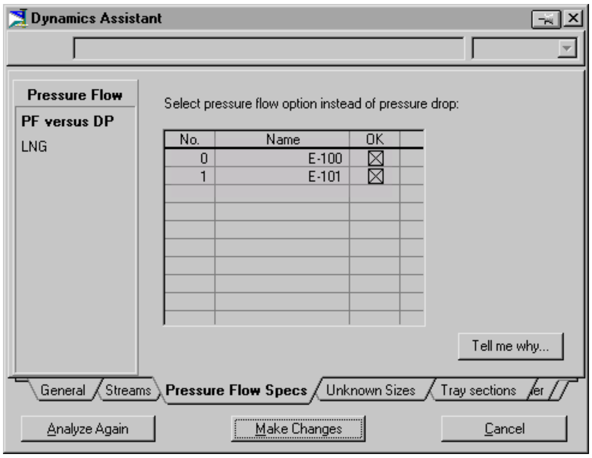


The **Internal Flow Specs** page lists the internal streams which require a flow specification. This is used primarily where separators are directly connected to each other by two or more streams. The flow specification which is added is sufficient to start a case in dynamics, however, it is highly recommended that a unit operation with pressure flow relationships (as such an operation is probably missing) be placed between such separators. Examples of unit operations with pressure flow relationships include: valve, compressor, pump, and heater. Once such a unit operation has been placed between the separators, the flow specification can be removed.

2.1.3 Pressure Flow Specs Tab

PF versus DP Page

Figure 2.11

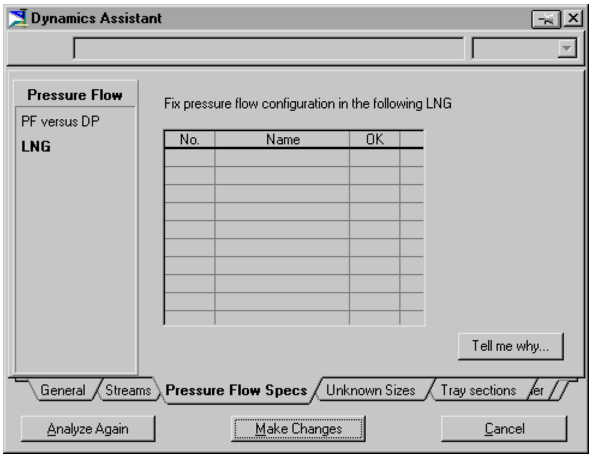


The **PF versus DP** page lists the unit operations which currently have a specified pressure drop as the dynamic specification. The pressure drop option should not be used because this is physically unrealistic. Material flow is driven by pressure differences as well as resistances and stops when the pressures have been equalized. A fixed pressure drop specification does not allow for this process. The k values are calculated based on the initial specified pressure drop.

If you do not want HYSYS to change the pressure drop specification to a pressure flow specification, simply uncheck the **OK** box for the unit operation. Double clicking on the name of the unit operation will open its property view.

LNG Page

Figure 2.12

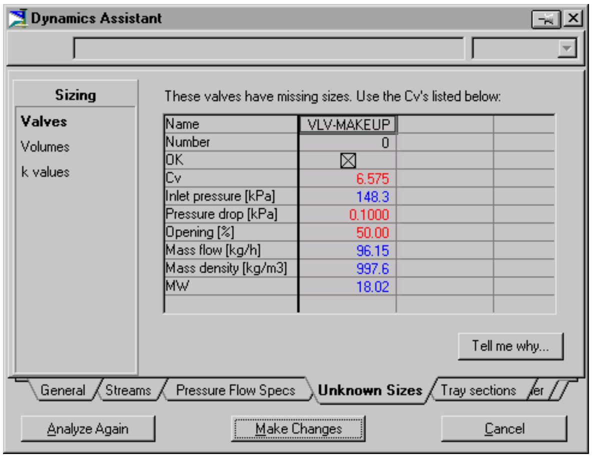


The **LNG** page indicates which LNG exchangers are currently specified with a pressure drop specification or which LNGs are missing k values (depending on the dynamic rating method chosen). Pressure drop specifications should be changed to either pressure flow equation specifications (k values) or suitable pressure drop correlations.

2.1.4 Unknown Sizing Tab

Valves Page

Figure 2.13



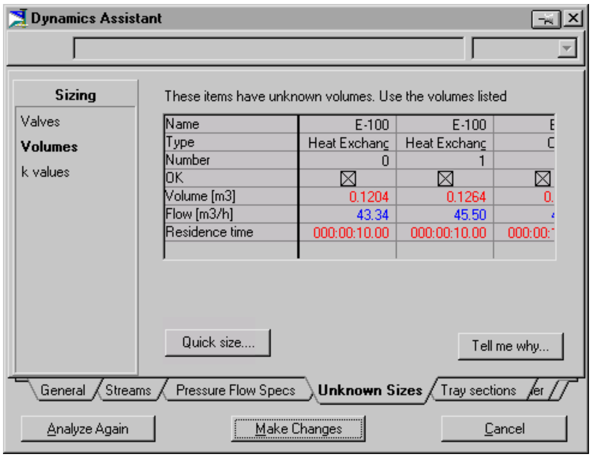
The **Valves** page lists the valves which are not sized. The current conditions for the valve are listed and the calculated valve Cv based on the pressure drop and percent opening of the valve, both of which may be changed directly on the page. By default, any valve Cv values which are calculated to be less than 50 are defaulted to 50.

It is possible to change any of the sizing data for the valve. The Cv value will be updated based on any changes that are made. If you change Cv, the new value will be added to the valve when the ***Make Changes*** button is pressed.

If you do not want HYSYS to size a valve, uncheck the **OK** box for the valve.

Volumes Page

Figure 2.14



The **Volumes** page lists the unit operations which have unknown volumes. Units which need volumes include: Separators (regular, 3 phase and tanks), condensers, reboilers, reactors, heat exchangers, air coolers, coolers, and heaters.

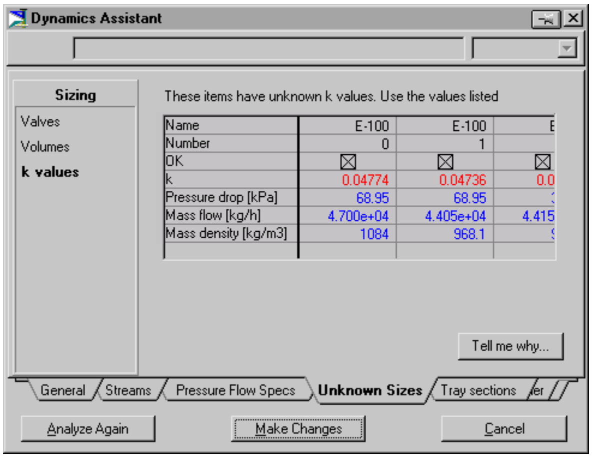
The unknown volumes are calculated based on the volumetric feed flowrate and the specified residence time. However, if the quick sizing feature is applicable, its volume and corresponding residence time will be given.

> If you do not like the suggested volume, either specify a new value or have it calculated by changing the volumetric flow rate and the residence time.

If you do not want HYSYS to implement the calculated volume for a unit operation, uncheck the **OK** box for the unit operation.

k values Page

Figure 2.15



*For more information on **k** values or other pressure flow parameters, consult **Chapter 1.3.2 - Basic Pressure Flow Equations**.*

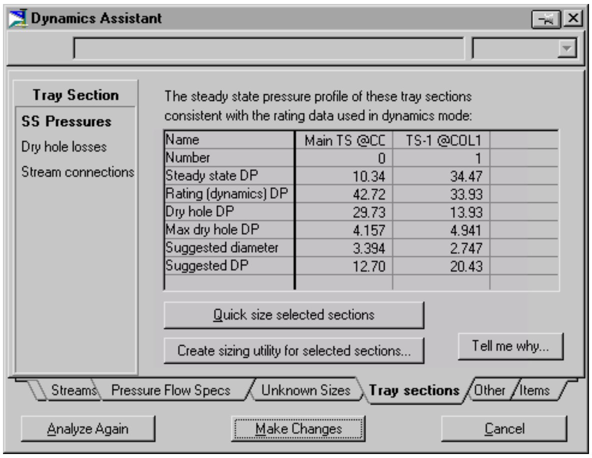
The **k values** page lists unit operations for which the *k* value is unknown. The information required to calculate the k value is listed along with the current calculated *k* value. Unit which can have '*k*' values include: heaters, coolers, heat exchangers, air coolers, and valves.

If you do not want HYSYS to calculate a *k* value, uncheck the **OK** box for the unit operation.

2.1.5 Tray Sections Tab

SS Pressures Page

Figure 2.16



*The **SS Pressure** page is active in Steady State mode only.*

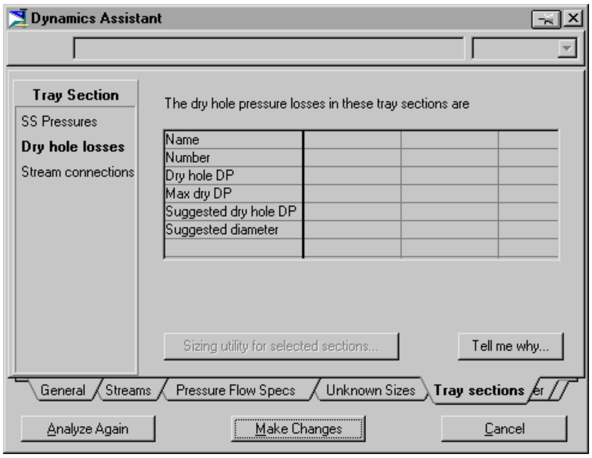
The **SS Pressure** page identifies tray sections where the total steady state pressure drop seems to be inconsistent with the total pressure drop calculated according to the dynamics rating model. By default, the Assistant provides tray section parameters based on its own internal sizing method. However, you may choose to use the ***Create sizing utility for selected sections*** button which brings up the Tray Sizing utility. The results from the utility can be exported to the tray sections. You can also invoke the quick size feature for selected sections and then repeat the analysis to check if that resolves problems.

If the dry hole pressure loss is larger than the suggested maximum, a diameter is suggested. If you use this value and calculate k values based on it, the dry hole pressure losses should be more realistic. The Dynamics Assistant make changes to the diameter such that the new pressure flow k values for the vapour will give reasonable pressure drops. The steady state pressure profile will be updated to the suggested value.

You can view the tray section property view by double click on the tray section name.

Dry Hole Losses Page

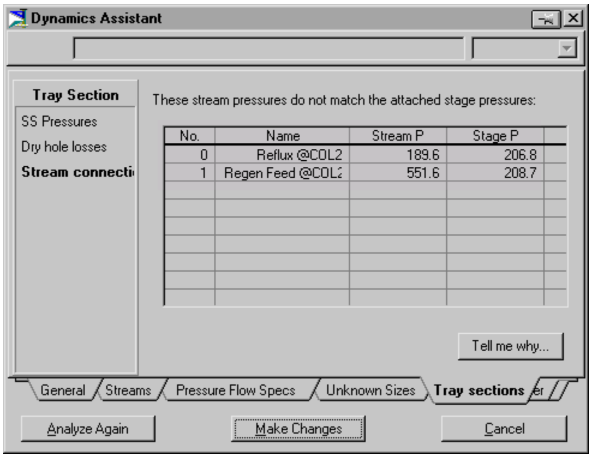
Figure 2.17



The **Dry hole losses** page displays the tray sections where the dry hole pressure loss is very high.By default the assistant fixes this by changing the diameter of the tray section and calculating new k values based on it. The ***Sizing utility for selected sections*** button opens the tray sizing utility view which can be used to minimize the pressure losses.

Stream Connections Page

Figure 2.18



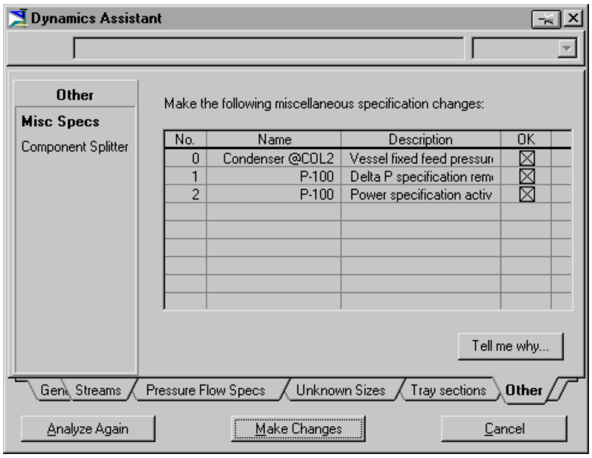
The **Stream connections** page list the streams whose pressure do not match with those of the trays where they connect. For a ratings model you should supply any missing valves or pumps that are needed in the steady state model.

This page is not active when static heads are enabled or in dynamics mode.

2.1.6 Other Tab

Misc Specs Page

Figure 2.19

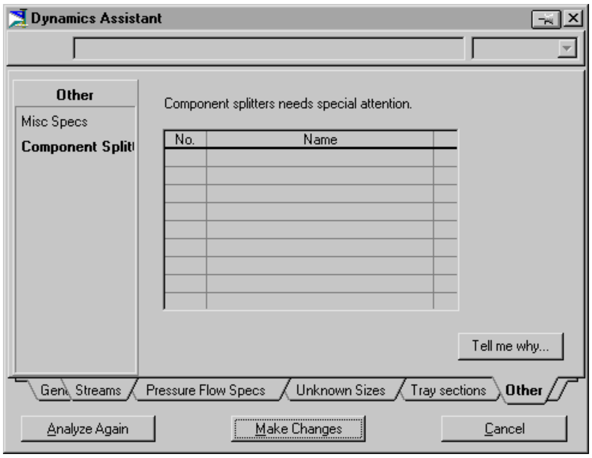


The **Misc** page displays the dynamic specifications which are not based on sizing equipment, adding valves, or adding pressure and flow specs. Some examples of the types of changes which may be made are listed below:

- Pump delta P specs removed
- Pump power spec activated
- Mixer equal pressure option active
- Compressor power spec activated
- Expander power spec activated

Component Splitter Page

Figure 2.20



The **Component Splitter** page display all the component splitters in your case. They do not follow pressure flow principles because they have fixed flow split fractions. Therefore, the Assistant can not properly cope with them.

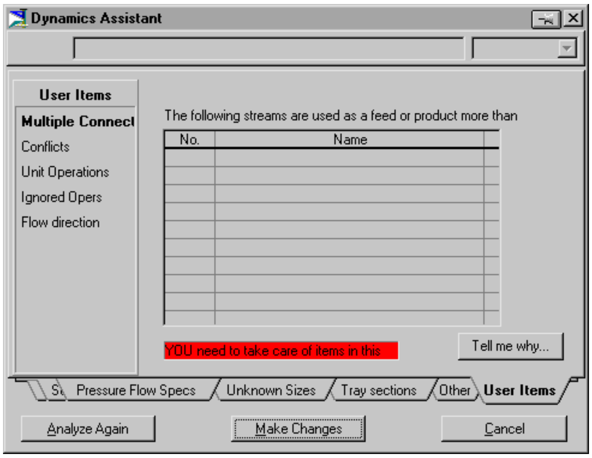
It is recommended that all connected streams have pressure specifications. Of the three branches (feeds, overhead product and bottoms product) all must have typical pressure flow specifications except two which must have one specification fewer each.

Where a component splitter is not connected to other operations all feed flow rates will be set to specifications with the two product flow rates not specified. Connected component splitters will need your intervention.

2.1.7 User Items Tab

Multiple Connections Page

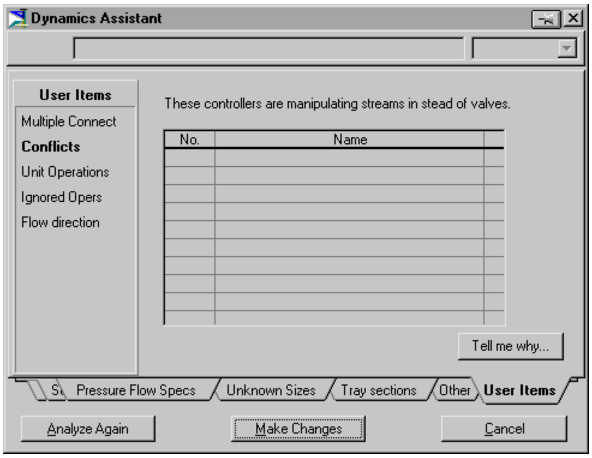
Figure 2.21



The **Multiple Connections** page lists any streams which are used in multiple connections. Streams which are attached as feeds or products to multiple unit operations are not allowed in dynamics since they represent physically impossible situations. You must make any changes listed here, the Dynamics Assistant does not correct any multiple connection errors. It is recommended to delete these streams and make proper reconnections.

Conflicts Page

Figure 2.22

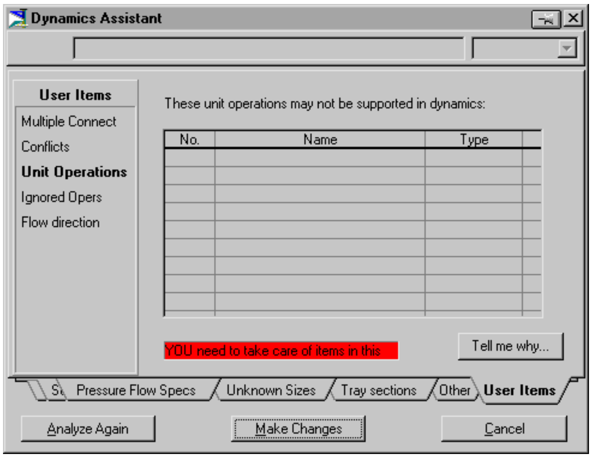


The **Conflicts** page lists any streams which have their flows directly controlled by controllers. It also lists streams which do not contain flow specifications or have specifications that the Assistant recommends you remove. Although this is allowed in HYSYS, it is recommended that a valve be placed in the stream to control the flow. If it is desired to directly control the flow of the stream another specifications will need to be disabled. This method can cause singular solution in the Flowsheet. The Assistant will fix this problem by moving the controller OP to a valve when one exists or is added.

You can open the property view for a given unit operation by double clicking on its name.

Unit Operations Page

Figure 2.23

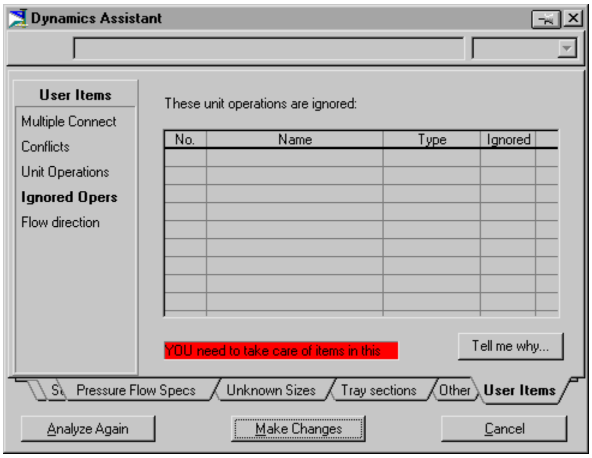


The **Unit Operations** page lists any unit operations in the case which are not supported in dynamics. These unit operations should be either deleted, replaced with a suitable unit operation which is supported in dynamics, or disconnected from the active flowsheet and ignored.

You can open the property view for a given unit operation by double clicking on its name.

Ignored Opers Page

Figure 2.24

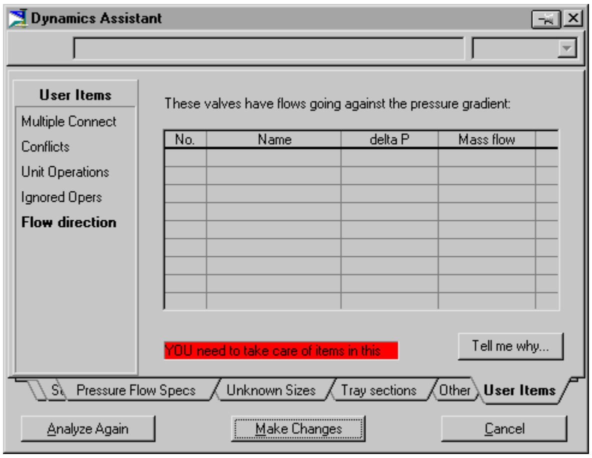


You can open the property view for a given unit operation by double clicking on its name.

The **Ignored Opers** page lists any unit operations which are currently ignored. You need to fix any problems listed on this page as the Dynamics Assistant cannot fix them. If an ignored unit operation should not be ignored, uncheck the ignored box for the unit operation. If you wish that a unit operation not be included in the dynamic simulation, *disconnect* it from the active flowsheet.

Flow Direction Page

Figure 2.25



*The **Flow direction** page is only active when static heads are disabled.*

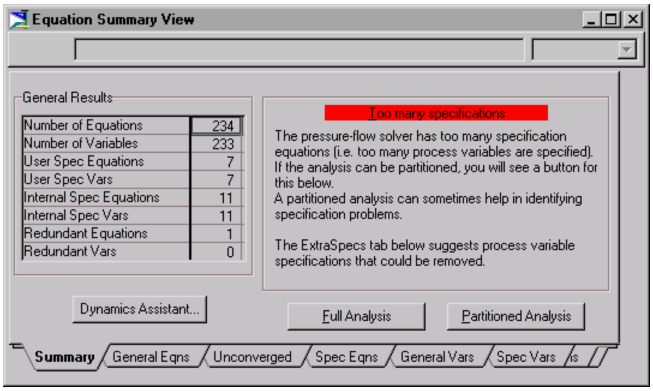
The **Flow direction** page lists the valves where the flow is going from low to high pressure or where the pressure drop is zero for non zero flow.

2.2 Equation Summary View

The **Equation Summary** view is accessed by selecting **Equation View Summary** from the **Simulation** menu.

2.2.1 Summary Tab

Figure 2.26



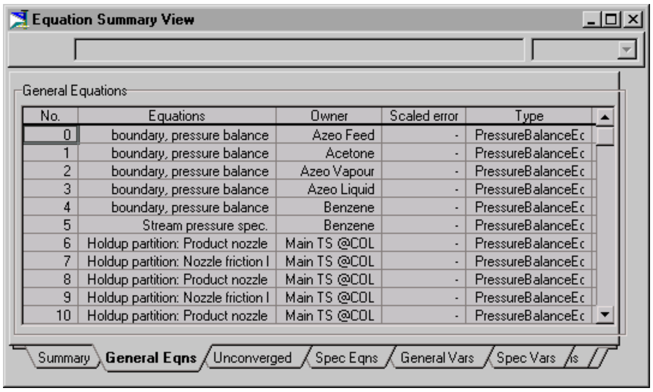
The **Summary** tab contains information which can help you in finding where there are specification problems in your case. The **General Results** group contains a summary of the number of equations and variables that are in the case. The group box to the right, contains information regarding the general status of the case. If there are problems with the specifications, some basic information is provided here. If you open **Equation Summary** view from the Menu Bar, the **Summary** tab initially contains a single button, ***Full Analysis***. Pressing this button causes HYSYS to analyze the pressure-flow parameters in order to determine if there are enough specifications for the problem. If HYSYS determines a problem, the ***Dynamics Assistant*** button will become visible and the ***Partitioned Analysis*** button may become visible. The ***Full Analysis*** and ***Partitioned Analysis*** use different methods of analyzing pressure-flow parameters.

In addition, should HYSYS detect any problems the **Unconverged** tab will be replaced with a **Extra Vars** or **Extra Specs** tab, depending on the nature of the specification problem.

2.2.2 General Equations Tab

The **General Eqns** tab contains a list of all the equations which will be used by the integrator. The number of equations corresponds to the value given in the **Number of Equations** cell on the **Summary** tab.

Figure 2.27

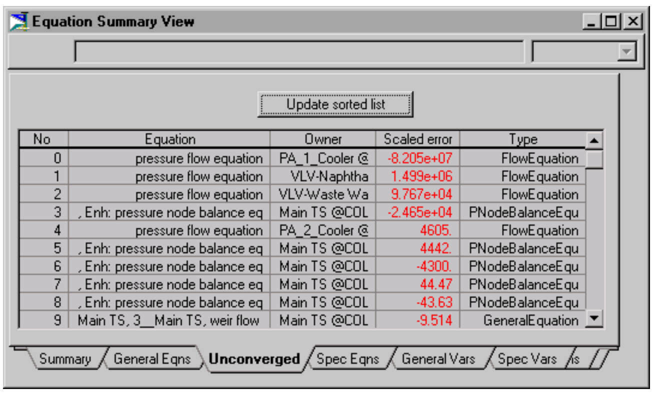


No.	Equations	Owner	Scaled error	Type
0	boundary, pressure balance	Azeo Feed	-	PressureBalanceEc
1	boundary, pressure balance	Acetone	-	PressureBalanceEc
2	boundary, pressure balance	Azeo Vapour	-	PressureBalanceEc
3	boundary, pressure balance	Azeo Liquid	-	PressureBalanceEc
4	boundary, pressure balance	Benzene	-	PressureBalanceEc
5	Stream pressure spec.	Benzene	-	PressureBalanceEc
6	Holdup partition: Product nozzle	Main TS @COL	-	PressureBalanceEc
7	Holdup partition: Nozzle friction l	Main TS @COL	-	PressureBalanceEc
8	Holdup partition: Product nozzle	Main TS @COL	-	PressureBalanceEc
9	Holdup partition: Nozzle friction l	Main TS @COL	-	PressureBalanceEc
10	Holdup partition: Product nozzle	Main TS @COL	-	PressureBalanceEc

Double clicking on the **Equations** cell for any given equation will open the property view for that equation. Double clicking on the **Owner** cell will open the unit operation or stream that the equation is attached to.

2.2.3 Unconverged Tab

Figure 2.28

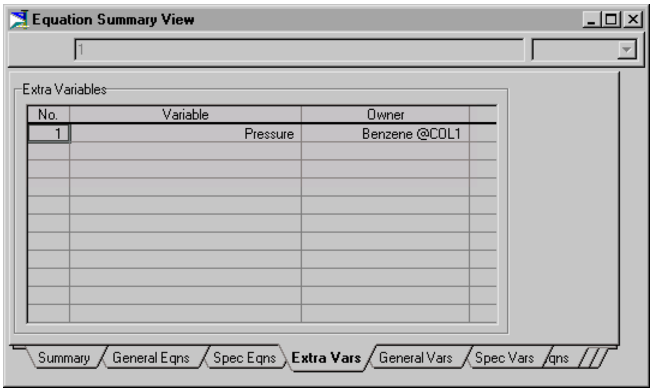


No	Equation	Owner	Scaled error	Type
0	pressure flow equation	PA_1_Cooler @	-8.205e+07	FlowEquation
1	pressure flow equation	VLV-Naphtha	1.499e+06	FlowEquation
2	pressure flow equation	VLV-Waste Wa	9.767e+04	FlowEquation
3	, Enh: pressure node balance eq	Main TS @COL	-2.465e+04	PNodeBalanceEqu
4	pressure flow equation	PA_2_Cooler @	4605.	FlowEquation
5	, Enh: pressure node balance eq	Main TS @COL	4442.	PNodeBalanceEqu
6	, Enh: pressure node balance eq	Main TS @COL	-4300.	PNodeBalanceEqu
7	, Enh: pressure node balance eq	Main TS @COL	44.47	PNodeBalanceEqu
8	, Enh: pressure node balance eq	Main TS @COL	-43.63	PNodeBalanceEqu
9	Main TS, 3__Main TS, weir flow	Main TS @COL	-9.514	GeneralEquation

If an error occurs while the integrator is running (Pressure Flow solver failed to converge) you can view the equations that had solved up to the point of failure. Pressing the ***Update Sorted List*** button causes HYSYS to reveal the type of equations, location, and scaled error associated with the unconverged nodes in the flowsheet. When trouble shooting your simulation, it is best to begin at the top of the equation list. These equations contain the largest error and therefore give the greatest insight into correcting the simulation.

2.2.4 Extra Variables Tab

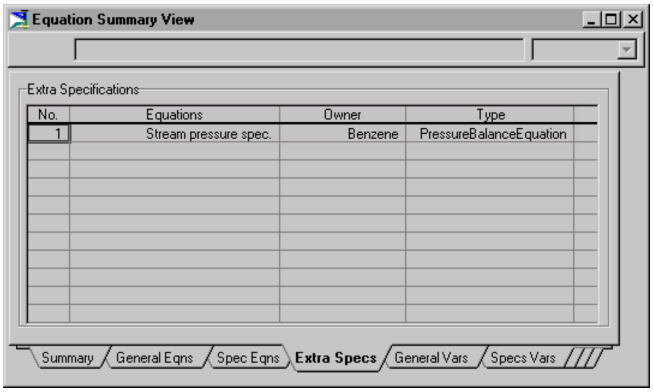
Figure 2.29



When the ***Full Analysis*** or ***Partitioned Analysis*** buttons are pressed, HYSYS determines that not enough specifications were known, and the **Extra Vars** tab is added to the **Equation Summary** view. This view shows possible variables that may be missing from the case. Double clicking on the **Variable** cell will open the **Variable** property view. Double clicking on the **Owner** cell will open the unit operation or stream that may require the missing variable.

2.2.5 Extra Specifications Tab

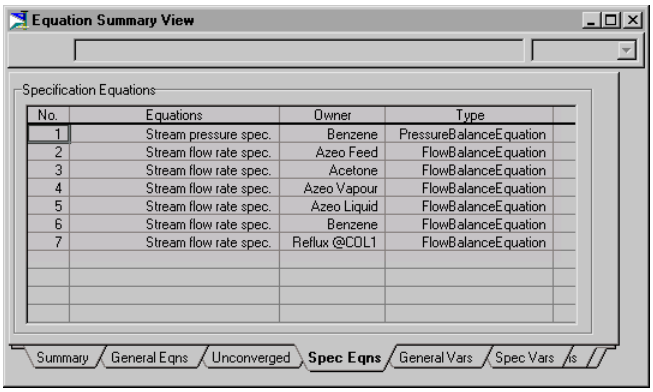
Figure 2.30



When the ***Full Analysis*** or ***Partitioned Analysis*** buttons are pressed, HYSYS determines that too many specifications were known, and the **Extra Specs** tab is added to the **Equation Summary** view. Double clicking on the **Equations** cell will open the property view for the equation. Double clicking on the **Owner** cell will open the unit operation or stream that may contain the extra specification.

2.2.6 Specified Equations Tab

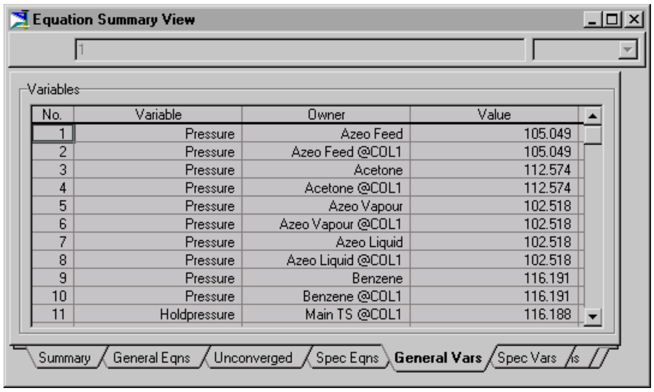
Figure 2.31



The **Spec Eqns** tab contains a list of the specified equations in the case. Double clicking on the **Equations** cell will open the property view for the equation, while double clicking on the **Owner** cell will open the property view for the unit operation or stream that uses the equation. The **Type** cell displays the type of specified equation such as: **Pressure Balance Equation** or **Flow Balance Equation.**

2.2.7 General Variables Tab

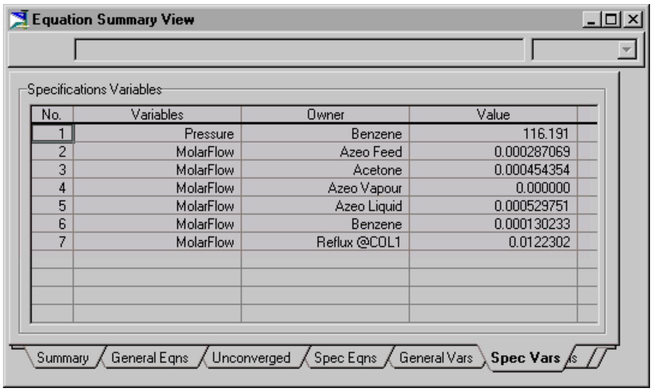
Figure 2.32



The **General Vars** tab contains the list of variables being used by the integrator for dynamic simulation. This number corresponds to the number displayed in the **Number of Variables** cell on the **Summary** tab. Double clicking on the **Variable** cell opens the property view for that particular variable.

2.2.8 Specification Variables Tab

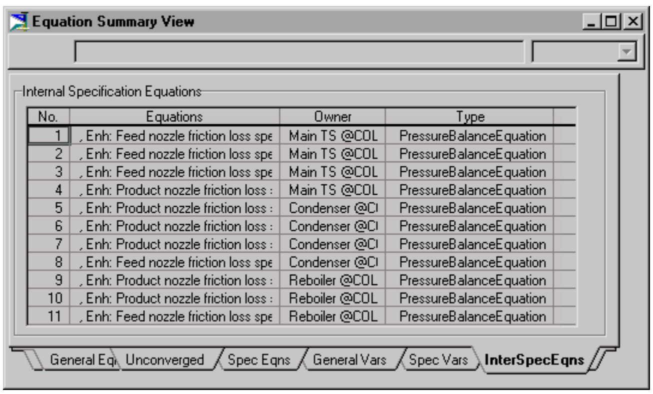
Figure 2.33



The **Specs Vars** tab contains the list of specified variables being used by the integrator for dynamic simulation. This number corresponds to the number displayed in the **User Spec Vars** cell on the **Summary** tab. Double clicking on the **Variable** cell opens the property view for the particular variable.

2.2.9 Internal Specification Equations Tab

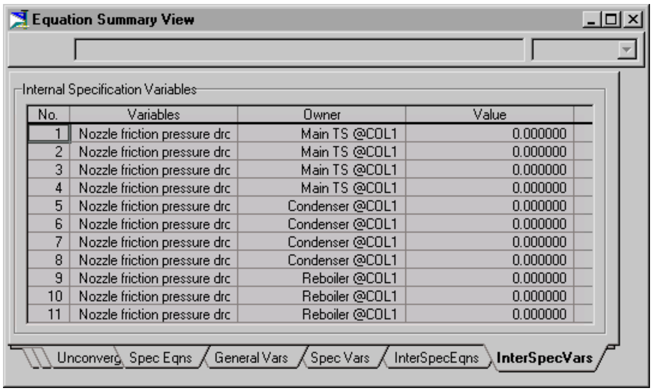
Figure 2.34



The **InterSpecEqns** tab contains the list of Internal Specification Equations which are used by the integrator for dynamic simulation. Double clicking on the **Equations** cell opens the property view for the equation, while double clicking on the **Owner** cell opens the unit operation or stream associated with the equation.

2.2.10 Internal Specifications Variables Tab

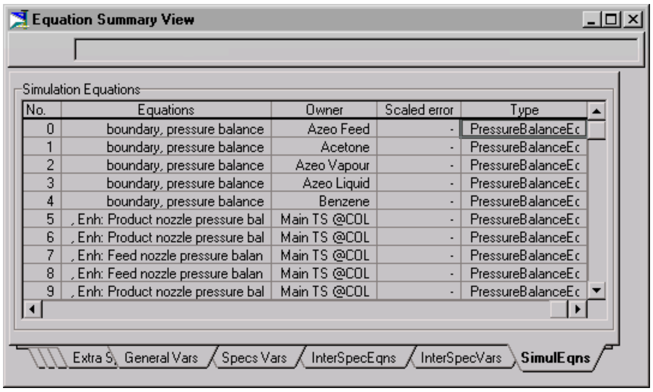
Figure 2.35



The **InterSpecVars** tab contains the list of Internal Specification Variables which are used by the integrator for dynamic simulation. Double clicking on the **Variables** cell opens the variable property view, while double clicking on the **Owner** cell opens the unit operation or stream from which the variable is taken.

2.2.11 Simultaneous Equations Tab

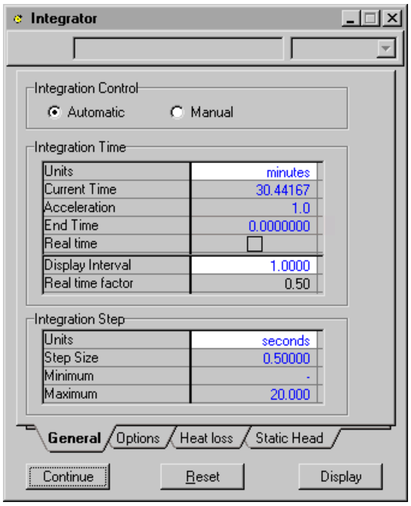
Figure 2.36



The Simultaneous Equations (**SimulEqns**) tab lists the equations that are solved simultaneously by the integrator. Double clicking on the **Equations** cell will open the property view for the equation, while double clicking on the **Owner** cell will open the property view for the unit operation or stream that uses the equation. The **Type** cell displays the type of specified equation such as: **Pressure Balance Equation** or **Flow Balance Equation.**

2.3 Integrator

Figure 2.37



The Integrator is used when running a case in **Dynamic** mode. You can access the Integrator view from the **Simulation** menu or by using the **CTRL I** hot key.

HYSYS solves all equations using the fully implicit Euler integration method. On the **Integrator** view, various integration parameters can be specified.

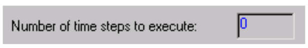
2.3.1 General Tab

The **General** tab has three group boxes which contain the time parameters for the integrator. The **Integration Control** group has the controls for whether the integration is **Automatic** or **Manual**. Manual integration lets you specify the number of time steps which HYSYS will execute. Once the integrator has executed the number of time steps, the integrator will not stop but remain in a holding mode. If additional time steps are entered, the integrator will continue integration for the given number of time steps.

Integration Time Group

The **Integration Time** group contains the following parameters:

Parameter	Description
Units	Selects the time units for the **Current Time**, **End Time**, and **Display Interval** fields.
Current Time	Displays the time that the Integrator has been running. When the Integrator is Reset, this value returns to zero. When the Integrator is not running this value can be changed to a user specified time.
Acceleration	If running in Real Time, changing this field can speed up (>1) or slows down (<1) the HYSYS model by talking larger or smaller steps (i.e. the factor is applied to the step size).
End Time	Allows you to specify the time at which the Integrator will stop.
Real Time	Forces the integration to proceed at real time. Note, if the real time factor is less than 1 (i.e. slower than real time) checking the **Real Time** box has no effect on the integration.
Display Interval	Only visible in **Automatic Integration Control**, this field contains the time interval at which HYSYS updates the views. The frequency of updating has a significant impact on the speed at which your simulation runs. The **Display Interval** has no effect on the calculation frequency.
Real time factor	Also only visible in **Automatic Integration Control**, this field is calculated by dividing a time interval for a case by the actual time required by HYSYS to simulate that time interval. The **Real time factor** depends on the computer's processing speed and the complexity of the simulation case.
Number of time steps to execute:	Number of time steps which HYSYS will execute. This field is only visible when in **Manual Integration Control**.



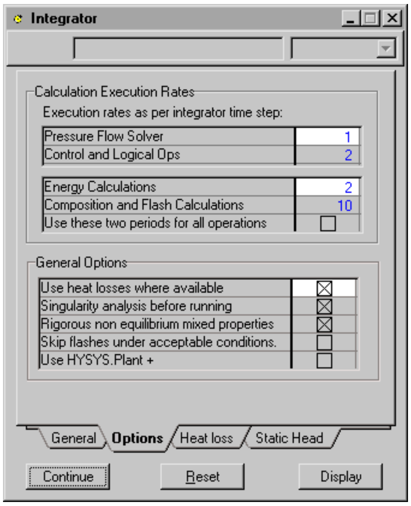
Integration Step Size Group

The **Integration Step Size** group contains:

Parameter	Description
Units	Allows you to select the units for the integration step size.
Step Size	Allows you to specify the integration step size, which by default is 0.5 seconds. While the integrator is running, this value cannot be changed.
Minimum	Allows you to specify the minimum number of steps that can be taken.
Maximum	Allows you to specify the maximum number of steps that can be taken.

2.3.2 Options Tab

Figure 2.38



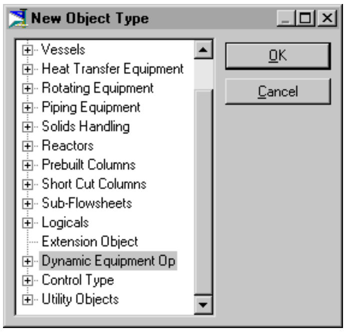
The **Options** tab contains the advanced parameters which may be used in dynamics. The **Execution rates as per integrator time step** group indicates the frequency at which the different balance equations are solved. The default values for **Pressure-Flow equations, Energy equations, Composition and Flash**, and **Control and Logic Ops** are **1, 2, 10**, and **2** respectively. A value of 2 for the Energy equations means that an energy balance will be performed every 2 time steps.

You have the option of specifying the composition and energy balance execution rates per integration time step for individual dynamic unit operations. In order to specify individual execution rates for different unit operations, it is necessary to add a new **Dynamic Equipment Ops** tab to the Workbook.

1. Deactivate the **Use these two periods for all unit operations** check box in the Calculation Execution Rates group box of the **Integrator** view.
2. Open the Workbook.
3. Select **Workbook** from the Menu Bar, and then select **Setup**. This opens the **Setup** dialog.

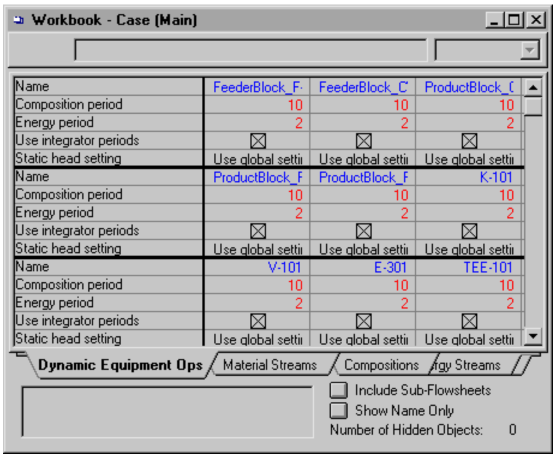
4. Select the ***Add*** button in the **Workbook Tabs** group. The **New Object Type** view is automatically opened.

Figure 2.39



5. Scroll down the list and select the **Dynamic Equipment Op** item. Press the ***OK*** button. You are returned to the **Setup** view, and the **Dynamic Equipment Op** variable set has been added as a **Workbook** tab.

Figure 2.40



6. Select the **Dynamic Equipment Op** tab in the **Workbook** and deactivate the **Use Integrator periods** check box for one of the unit operations. All other dynamic unit operations in the flowsheet will be deactivated, as well.

You may now specify individual execution rates the different unit operations.

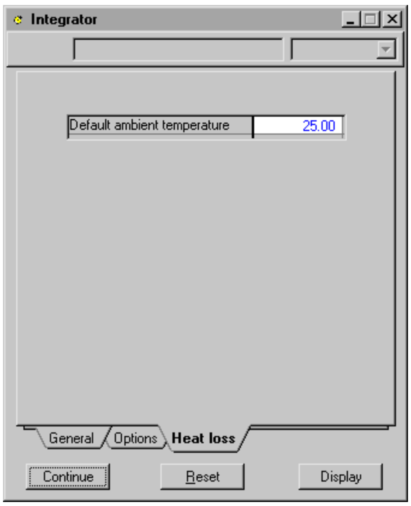
The **General Options** group contains the following parameters:

Parameter	Description
Use heat losses when available	When checked, heat loss models for holdup nodes will be accounted for in the energy balances. If not checked, all heat losses will be zero irrespective of individual heat loss settings.
Singularity analysis before running	When checked, HYSYS warns you of a possible singular solution matrix before starting integration. For larger cases it is recommended that this option be deactivated to increase the overall start up speed. For cases where a singular solution is not considered to be a problem, this option can be disabled which will increase the overall speed.
Rigorous non equilibrium mixed Properties	It is recommended that this option remains active. Deactivating this option will provide a slight speed increase when nozzle efficiencies are not 100%, although instabilities may occur.
Skip flashes under acceptable conditions	It is recommended that this option remain in-active. Activating this option will tell HYSYS to skip flashes calculations under acceptable conditions. This will provide a slight speed increase, although instabilities may occur.
Use HYSYS.Plant +	When checked, the Plant + features are activated. Once the features have been used this option can not be deactivated.

2.3.3 Heat Loss Tab

The **Heat Loss** tab allows you to specify the ambient temperature that is used in the heat loss equations.

Figure 2.41

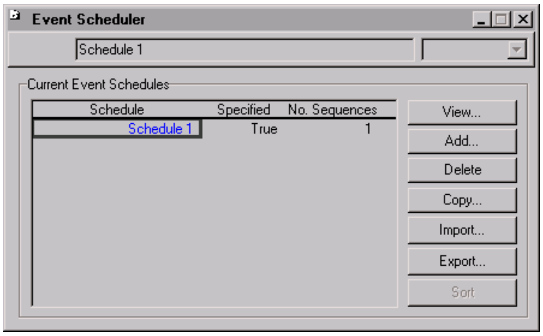


2.3.4 Static Head Tab

The options on this tab requires the HYSYS.Plant+ feature. Refer to **Section 1.5 - Plant+** for more information on Plant+ options.

2.4 Event Scheduler

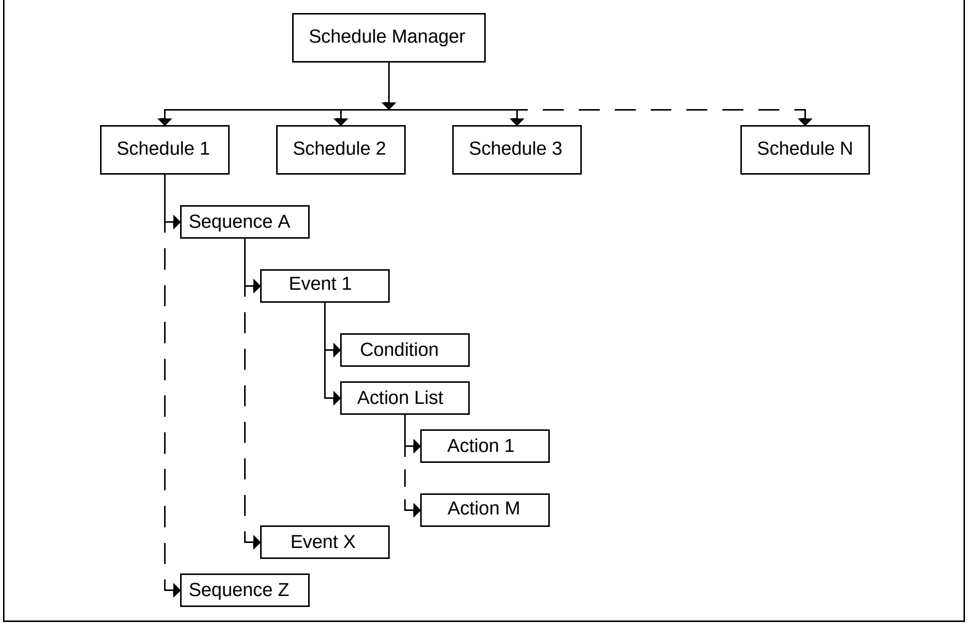
Figure 2.42



Using the Event Scheduler, it is possible to have HYSYS perform given tasks at predetermined times once a simulation is running in dynamics. The tasks can be triggered by a pre-determined simulation time, a logical expression becoming true, or a variable stabilizing to within a given tolerance for a set amount of time.

The Event Scheduler Manager contains all the Event Schedules in the current HYSYS case. Each Schedule is comprised of Sequences, which in turn are made up of Events. An Event must have a Condition, which determines when the event will occur, and at least one enabled action to be fully defined.

Figure 2.43

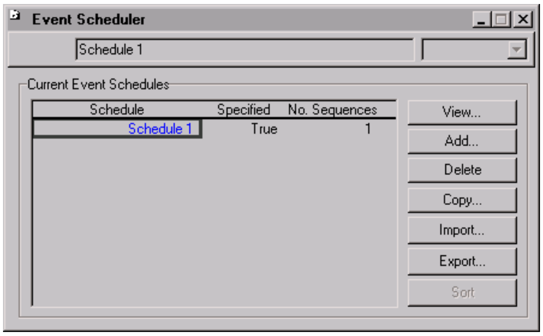


The **Event Scheduler** is accessed from the **Simulation** Menu Bar, or by using the hot key **CTRL E**.

2.4.1 Event Scheduler Manager

When you open the **Event Scheduler**, HYSYS displays the **Event Scheduler Manager**, which contains a list of the Schedules in the case.

Figure 2.44



The buttons in the **Current Events Schedule** group are used to manage the schedules for the current case:

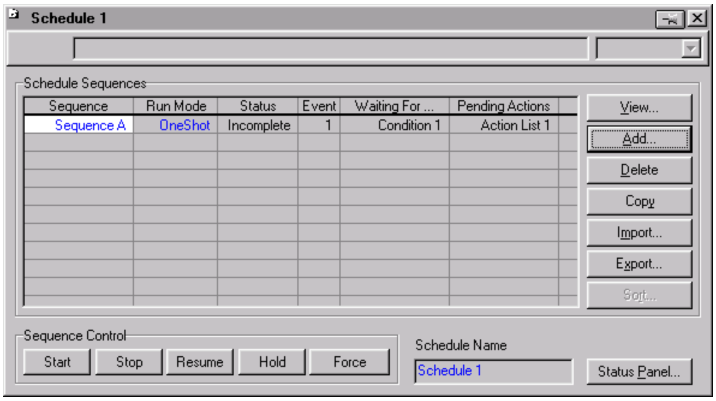
Button	Description
View	Allows you to view the selected schedule. This is only active when a schedule exists in the case.
Add	Allows you to add new schedules to the case.
Delete	Allows you to delete the selected schedule. This is only active when a schedule exists in the case.
Copy	Allows you to make a copy of the selected schedule. This is only active when a schedule exists in the case.
Import	Allows you to import a saved schedule from disk. Schedules have the extension *.sch.*
Export	Allows you to export the selected schedule to disk. Once exported, a schedule can be retrieved using the Import button. This is only active when a schedule exists in the case.
Sort	Allows you to arrange the schedules. This is only active when at least 2 schedules exist in the case.

The **Specified** field in the matrix indicates whether all the Events in the Schedule have been fully specified. If all the Events in all the Sequences which make up the Schedule have been fully specified, the **Specified** field will display **True**, otherwise it will be **False**. The **No. Sequences** field displays the number of Sequences in the Schedule.

2.4.2 Schedule View

When you press the ***Add*** button from the **Event Schedule Manager**, HYSYS opens a **Schedule** view.

Figure 2.45



In the **Schedule Sequences** group, there are seven buttons which allow you to organize all the Sequences for the current Schedule.

Button	Description
View	Allows you to view the selected sequence. This is only active when a sequence exists in the schedule.
Add	Allows you to add new sequence to the schedule.
Delete	Allows you to delete the selected sequence. This is only active when a sequence exists in the schedule.
Copy	Allows you to make a copy of the selected sequence. This is only active when a sequence exists in the schedule.
Import	Allows you to import a saved sequence from disk. Schedules have the extension *.seq*.
Export	Allows you to export the selected sequence to disk. Once exported, a sequence can be retrieved using the Import button. This is only active when a sequence exists in the schedule.
Sort	Allows you to arrange the sequences. This is only active when at least 2 sequences exist in the schedule.

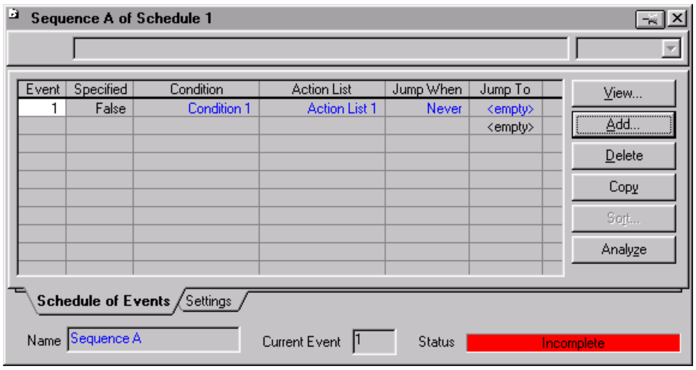
The **Sequence** field contains the name of the Sequence. The **Run Mode** field displays the sequence mode; **One Shot** or **Continuous**. A **One Shot** Sequence executes all its Events in order then its **Status** changes to

Complete. A **Continuous** Sequence returns to the first Event after the last Event has been executed, in a continuous loop. The **Event**, **Waiting For**, and **Pending Actions** fields display the current event number with its corresponding Condition name and Action List name.

2.4.3 Sequence View

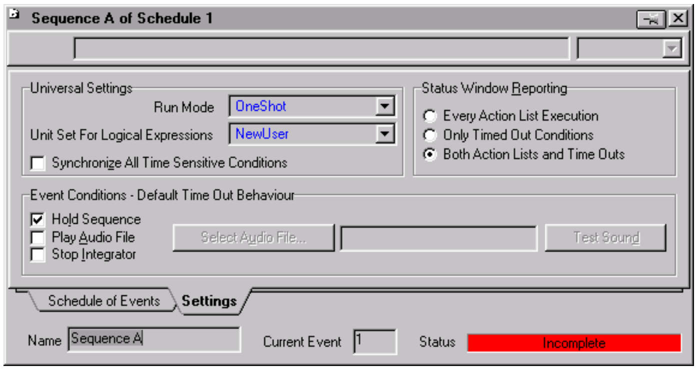
When you press the ***Add*** button from the **Schedule** view, the **Sequence** view opens.

Figure 2.46



The **Schedule of Events** tab shows the list of events for the schedule. The **Specified** field in the matrix indicates whether the event has been fully defined. The **Condition** field displays the condition name of the event, while the **Action List** field shows the name of the events action list. If the event is not fully specified the ***Analyze*** button is enabled and provides additional feedback. The **Jump When** field displays whether the Event will Jump over any Events and if so under what condition. The default value is Never and is configured on the **Branching & Time Out Behaviour** tab of the **Event** view. The **Jump To** field will display the event to jump to. In this way a single Event or a group of events can be skipped under certain circumstances. At the bottom of the view the sequence name, current event number and sequence status are shown.

Figure 2.47

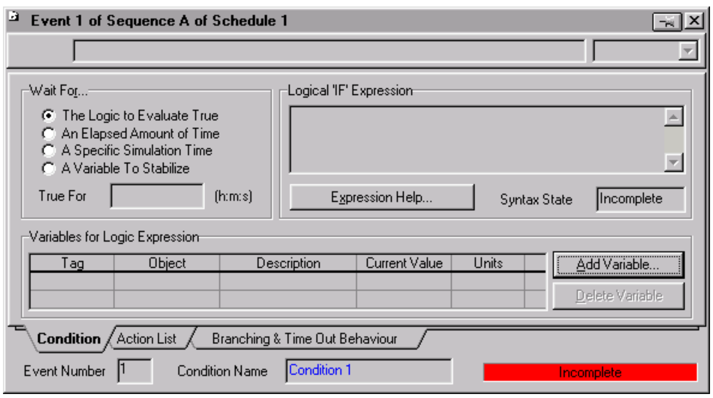


On **Settings** tab, the sequence **Run Mode** is selected, a unit set global to the sequence can be chosen and different Status Window Reporting options are available. Selecting the **Synchronize All Time Sensitive Conditions** check box will assure execution of a particular event at an exact simulation time. This applies only when one of the two time conditions has been selected for the Event. Default Time Out Behaviour is specified here and applies to all the events unless a specific event overrides the behaviour.

2.4.4 Event View

When you press the ***Add*** button from the **Schedule of Events** tab of the **Sequence** view, HYSYS opens the **Event View** on the **Conditions** tab.

Figure 2.48



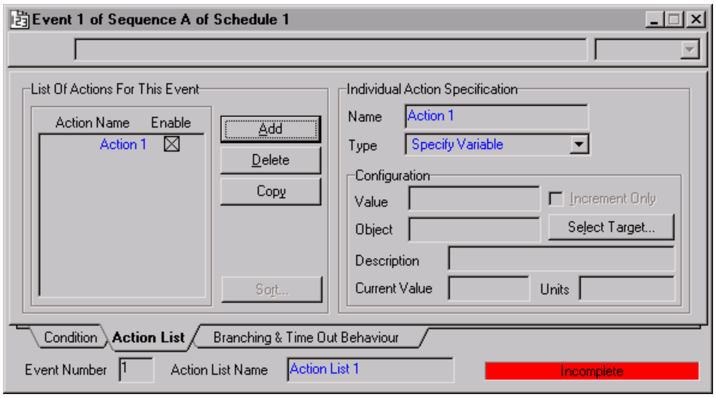
The **Conditions** tab shows the four possible conditions located within the **Wait For** group box.

Condition	Description
The Logic to Evaluate True	Requires variables whose Tags will become part of the **Logical 'If' Expression**. Variables are selected by pressing the ***Add Variable*** button, which opens the Variable Navigator. If no expression is provided the condition evaluates **True**. Help is available by pressing the ***Expression Help*** button. This is the same logic that is used by the Spreadsheet. The **Syntax State** reports whether or not the expression is valid syntax and returns **Incomplete**, **Bad Syntax**, or **Complete** not the evaluation of the expression. Optionally a time value may be input in the **True for** field, in which case the expression must evaluate **True** for this length of time before executing the associated actions.
An Elapsed Amount of Time	The actions execute after this amount of time has lapsed.
A Specific Simulation Time	Actions execute only at the specific simulation time.
A Variable to Stabilize	The actions execute once a variable has stabilized. In order for the variable to stabilize the following information is required: • a basis for calculation • a bandwidth about the basis • a period that the variable must be within the bandwidth Two generalized situations, monotonic increase or decrease (i.e. 1st order response) or oscillating increase or decrease (i.e. underdamped 2nd order response), are encountered. The user inputs are a variable of interest (i.e. Feed Pressure), variable bandwidth expressed as either a percentage (i.e. ±2.5% of the variable average over the period) or an absolute variation of the variable (i.e. ±2kPa) and a period that the variable must be within the given bandwidth. The condition evaluates **True** when the selected variable value is within the given bandwidth for the specified length of time.

Action List Tab

On this tab, the individual actions for the Event are added. Press the ***Add*** button to create a new Action.

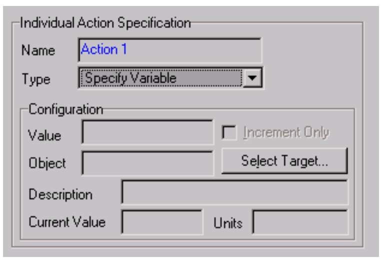
Figure 2.49



The name and type of the action selected in the **List of Actions for This Event** group can be changed in the **Individual Action Specification** group. The **Name** cell allows you to change the action name while the **Type** drop down provides a list of the available action types. Once an action type has been chosen, the **Configuration** group will change to show the information required for the action type.

Specify Variable

Figure 2.50

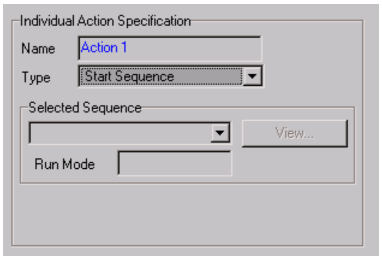


The **Specify Variable** action requires an Object and a Value. To select an Object, press the ***Select Target*** button which opens the Variable Navigator. The current value and units of the selected object are shown.

If the **Increment Only** check box is selected the **Value** cell is added to the **Current Value** cell at the time of execution otherwise the **Value** cell replaces the **Current Value** cell.

Start/Stop/Hold/Resume Sequence

Figure 2.51

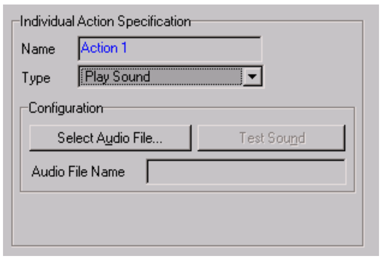


The **Start**, **Stop**, **Hold** and **Resume Sequence** actions all require that a Sequence is selected from the drop down list. The ***View*** button displays the selected sequence. Note that it is not possible for a Sequence to Start or Resume itself, but it can Stop itself and put itself in Hold Mode.

Note that the available Sequences are only those that are part of the parent Schedule.

Play Sound

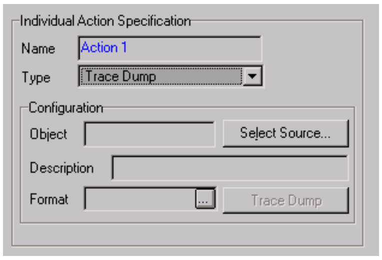
Figure 2.52



The **Play Sound** action requires a *.wav* audio file, which can be selected by pressing the ***Select Audio File*** button. When a file has been selected the ***Test sound*** button is enabled and the audio file can be played.

Trace Dump

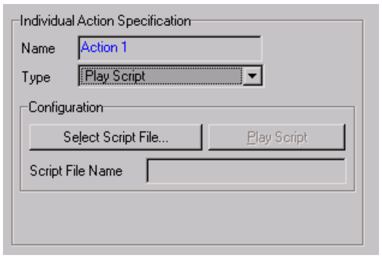
Figure 2.53



The **Trace Dump** action requires the selection of a source variable by pressing the ***Select Source*** button. The default format for the variable is shown, and may be changed by pressing the button at the end of the **Format** cell. When an object has been selected, the ***Trace Dump*** button is activated, allowing you to dump the current variable information to the Trace Window.

Play Script Action

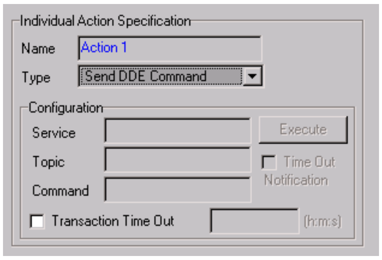
Figure 2.54



The **Play Script** action requires a HYSYS script file which can be selected by pressing the ***Select File*** button. When a file has been selected the ***Play Script*** button is enabled and the script can be played.

Send DDE Command

Figure 2.55



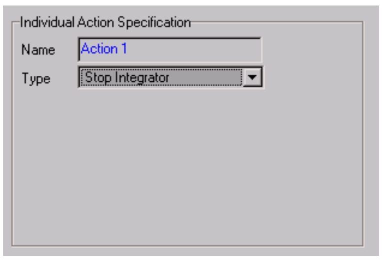
The **Send DDE Command** action requires inputs of Service, Topic, and Command. An example would be:

Service	Excel
Topic	Sheet1
Command	MyMacro

When execution occurs the Excel Sheet1 MyMacro will run using the Dynamic Data Exchange protocol. When the three inputs have been specified the ***Execute*** button becomes enabled and the action can be tested. In addition, the **Transaction Time Out** check box can be selected which then requires an input time. The command execution by the external service (i.e. Excel) will return control to HYSYS after the input time expires otherwise it waits until the external command completes its execution. When the **Transaction Time Out** check box is selected the **Time Out Notification** check box becomes enabled, and selecting this box will notify you if a time out occurs.

Stop Integrator

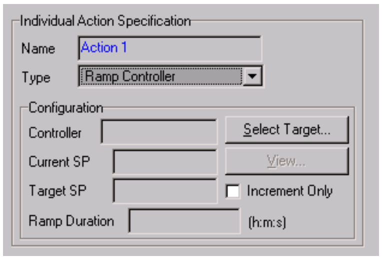
Figure 2.56



The **Stop Integrator** action stops the integrator.

Ramp Controller

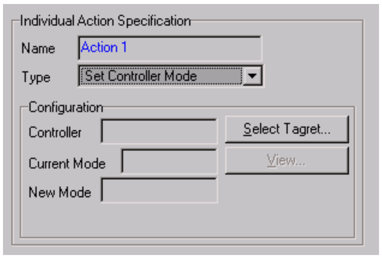
Figure 2.57



The **Ramp Controller** action requires a Controller, a Target Set Point and a Ramp Duration as inputs. The controller is selected by pressing the ***Select Target*** button that browses all the controllers in the case. After selecting a controller the current set point is shown for reference and the ***View*** button is enabled. Press the ***View*** button to open the **Controller** property view. Optionally, the **Increment Only** check box can be selected which increments the set point rather than setting it. Upon execution the controller is switched to Ramp mode, the target set point and duration are specified and the Ramp Controller message is sent.

Set Controller Mode

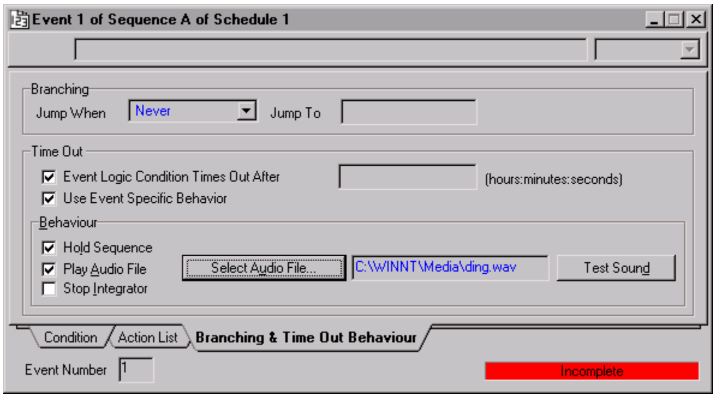
Figure 2.58



The **Set Controller Mode** action requires a Controller and a New Mode as inputs. The controller is selected by pressing the ***Select Target*** button that browses all the controllers in the case. After selecting a controller the current mode is shown and the ***View*** button is enabled.

Branching & Time Out Behaviour Tab

Figure 2.59



The information contained on this tab is not necessary for an event to be completely defined. By default, the **Jump When** condition for an event is set to **Never**, and the condition logic is not set to time out after a given time.

The drop down list in the **Jump When** cell contains four conditions:

Condition	Description
Never	The Event will not jump to another Event.
Always	The Event will always jump to another Event once it has executed.
True	The Event will jump to another Event once the Logic on the Condition tab is True.
Timeout	The Event will jump to another Event if the Timeout condition is met.

If a **Jump When** condition other than **Never** is chosen an Event must be chosen in the **Jump To** cell. The **Jump To** cell contains a drop down list with all the Events in the Sequence. Events can only jump to other Events in the same Sequence.

Initially, there is only one active option in the **Time Out** group. If you check the **Event Logic Condition Times Out After** check box, the **Use Event Specific Behaviour** check box will become active. Once the **Event Logic Condition Times Out After** check box is activated a value for the timeout must be specified in the cell directly to the right. The time entered will be the time when the Event Logic and thus the event will timeout. This facility allows for another OR in the Event condition logic.

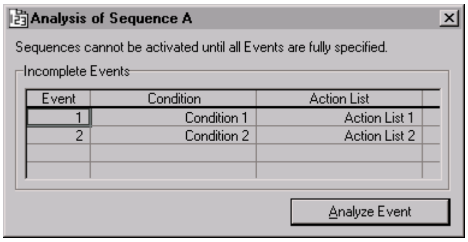
Checking the **Event Specific Behaviour** box will activate the **Behaviour** group. The three choices in the group can be activated in any combination from all three to none. These selections override the default behaviour defined in the parent sequence.

2.4.5 Analyzing a Schedule

HYSYS has a built in tool for analyzing Schedules which are not yet fully defined. From the **Sequence** view, it is possible to analyze an incomplete sequence by pressing the ***Analyze*** button; which is only active when the Sequence is incomplete.

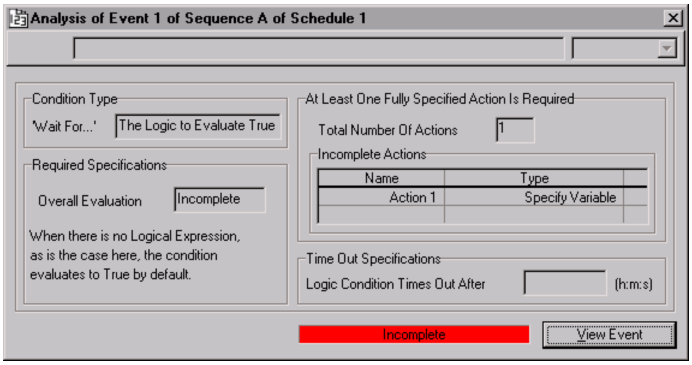
When the ***Analyze*** button is pressed, HYSYS opens the **Sequence Analysis** view, which is Modal.

Figure 2.60



This view displays a matrix with a list of all the Events that are not fully specified. Selecting an Event and double clicking in the **Event** column or pressing the **Analyze Event** button will open the **Event Analysis** view.

Figure 2.61



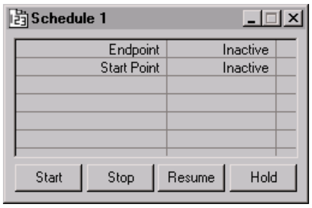
The **Event Analysis** view provides feedback about the Event. The contents of the **Required Specifications** group will change depending on the Event **Wait For** condition. Fields that are <empty> can be specified at this point or the ***View Event*** button can be pressed, opening the **Event** view where the specifications can be made. If the **Total Number of Actions** is zero, at least one fully specified and enabled Action must be provided. Actions that are not fully specified will be displayed in the **Incomplete Actions** matrix. Double clicking the **Name** field in the matrix will open the Event on the **Action List** tab with the Action selected.

Opening the an Event from the **Event Analysis** view will open a Modal Event view. If you press the ***Pin*** button to make the view non-Modal, the active view will return to the **Event Analysis** view which is also Modal.

2.4.6 Running a Schedule

After a Schedule has been fully defined, each Sequence will be in an Inactive state. Sequences can be run either from the Schedule Main view or from the Schedule Status panel shown below.

Figure 2.62



To activate a Sequence, select it from the list and press the ***Start*** button. Note the status changes from **Inactive** to **Waiting**. Of course the integrator has to be running for evaluations to occur. At every time step, any Sequences that are in the **Waiting** state will have their current *Event's Wait For Condition* evaluated in the order they appear in the list. When a current Event evaluates **True**, the associated Action List items are executed, and provided no Branching behaviour has been specified, the next Event in the list becomes the current event and evaluations continue. If the **Sequence RunMode** is **One Shot** and the last Event in the list executes the status changes from **Waiting** to **Complete** and the Sequence is reset.

2.4.7 Event Scheduler Example

To illustrate the use of the **Event Scheduler** a simple Schedule consisting of two sequences will be created. The goal will be for the first sequence to double the flowrate of a separator feed stream and then start the second sequence. The second sequence will then stop the integrator.

1. Create a fluid package using the **Peng Robinson** EOS and the following components: i-C4, n-C4, i-C5, n-C5, C6, and C7.

Property Package	Components
Peng Robinson	i-C4, n-C4, i-C5, n-C5, C6, C7

2. Install the stream **Feed** and define it as follows:

Name	Feed
Temperature [°C]	150
Pressure [kPa]	1500
Molar Flow [kgmole/hr]	100
Comp Mole Frac [i-C4]	0.6
Comp Mole Frac [n-C4]	0.01
Comp Mole Frac [i-C5]	0.01
Comp Mole Frac [n-C5]	0.01
Comp Mole Frac [C6]	0.1
Comp Mole Frac [C7]	0.27

3. Install a **SEPARATOR** unit operation with the following information:

UNIT OPERATION TYPE	SEPARATOR	
Tab [Page]	**Input Area**	**Entry**
Design [Connections]	Feed	Feed
	Vapour Outlet	Sep Vap
	Liquid Outlet	Sep Liq
Rating [Sizing]	Vessel Shape	Cylinder
	Vessel Orientation	Vertical
	Volume [m^3]	16
	Diameter [m]	1.8

4. Install a **VALVE** and define it as shown below:

UNIT OPERATION TYPE	VALVE	
Tab [Page]	**Input Area**	**Entry**
Design [Connections]	Feed	Sep Vap
	Product	Vap Prod
Rating [Sizing]	Pressure Drop [kPa]	10

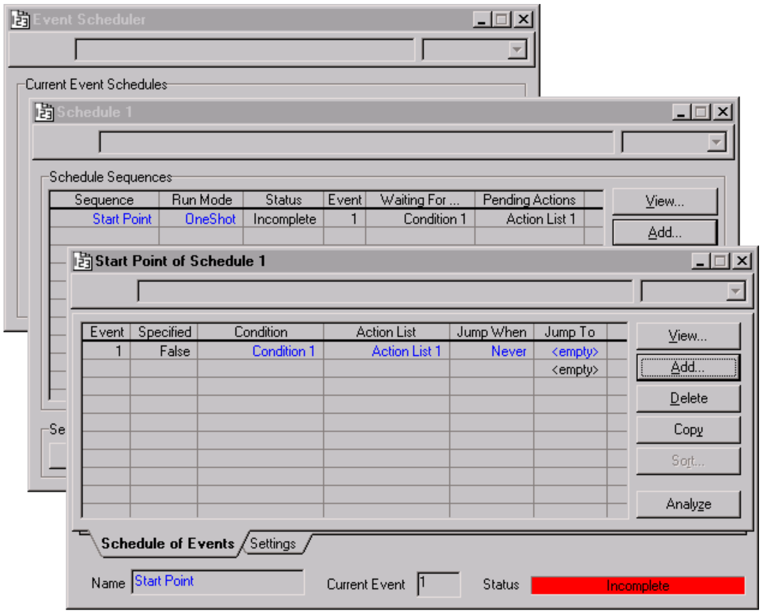
5. Press the ***Size Valve*** button to size the valve.
6. Install a second **VALVE** with the following information:

UNIT OPERATION TYPE	VALVE	
Tab [Page]	**Input Area**	**Entry**
Design [Connections]	Feed	Sep Liq
	Product	Liq Prod
Rating [Sizing]	Pressure Drop [kPa]	10

7. On the **Ratings** tab of this Valve's property view, press the ***Size Valve*** button to size the valve.
8. On the **Dynamics** tabs of their respective stream property views, assign Pressure specifications to streams **Liq Prod** and **Vap Prod**.
9. Assign a Flow specification to the **Feed** stream. Remove the Pressure specification.
10. Switch to **Dynamic** mode.
11. Open the **Event Scheduler.**
12. On the **Ratings** tab of this Valve's property view, press the ***Add*** button to create a new Schedule.

13. On the **Schedule** view, press the ***Add*** button to create a new Sequence.

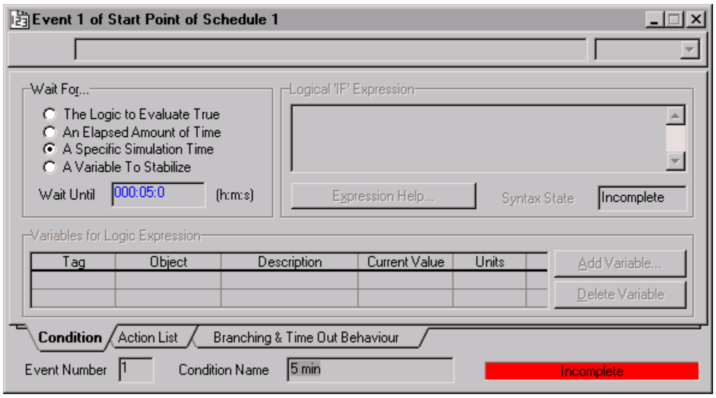
Figure 2.63



14. In the **Name** cell, change the Sequence name from **Sequence A** to **Start Point**.
15. On the **Sequence** view, press the ***Add*** button to create a new Event.
16. On the **Condition** tab of the **Event** view, select **A Specific Simulation Time** from the **Wait For** group.
17. Enter a time of **5 minutes** in the **Wait Until** cell.

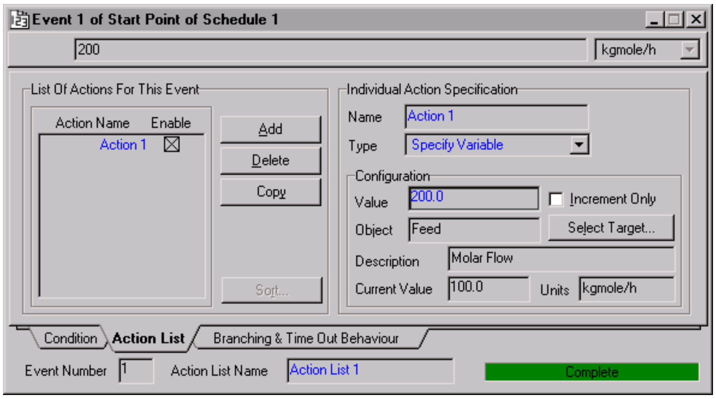
18. Change the **Condition** name to **5 min**.

Figure 2.64



19. On the **Action List** tab, press the ***Add*** button to create a new Action.
20. The default Action type is **Specify Variable** which is correct for this action.
21. Press the ***Select Target*** button and select the **Molar Flow** of **Feed** as the Object.
22. In the **Value** cell enter a value of **200 kgmole/hr**.
23. The **Status** of the Event will change from **Incomplete** to **Complete**.

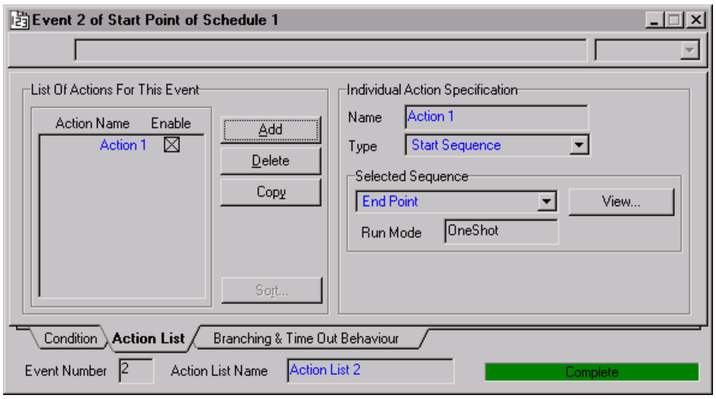
Figure 2.65



24. Close the **Event** view.
25. The current view should once again be the **Sequence** view. Close this view to return to the **Schedule** view.

26. Activate the cell below the Start Point cell. Press the ***Add*** button to add a second Sequence and rename it to **End Point**.
27. Press the ***Add*** button to create an Event.
28. For this Event on the **Conditions** tab, select **A Specific Simulation Time** of **10 minutes**.
29. Change the **Condition** name to **10 min**.
30. On the **Action List** tab, add an action and select **Stop Integrator** as the Action Type.
31. Close the views and return to the **Schedule** view.
32. Open the Sequence **Start Point** by either double clicking on the Start Point cell or by selecting the Start Point cell and then pressing the ***View*** button.
33. Activate the cell below Event 1 and press the ***Add*** button to add a second Event to the Sequence.
34. Change the **Condition Name** of the Event to **7 min**.
35. For the Event **7 min**, select **A Specific Simulation Time** of **7 minutes**.
36. Add an Action of type **Start Sequence**.
37. Select the Sequence **End Point**.

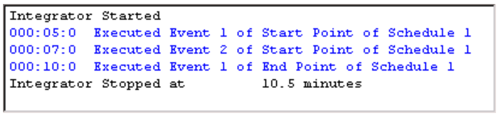
Figure 2.66



38. Close the views returning to the **Schedule** view.
39. Press the ***Status Panel*** button.
40. Select the Sequence **Start Point** and press the ***Start*** button. The **Status** of **Start Point** should change from **Inactive** to **Waiting** on both the Status Panel and the Schedule View.

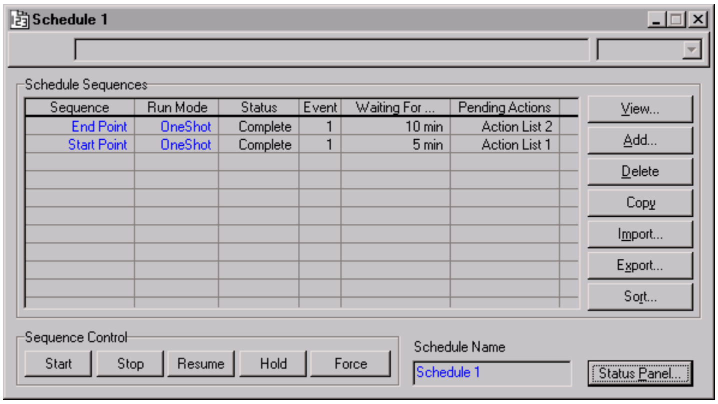
41. Start the **Integrator** and let it run.

Figure 2.67



If you watch the **Trace Window** and the **Status Panel** you will see the results in progress. At the 5 minute simulation time, a message will appear in the Trace Window stating that *Event 1* of **Start Point** has executed. At the 7 minute simulation time, a message will appear stating that *Event 2* of **Start Point** has executed. At this time, the Status of **End Point** will change from **Inactive** to **Waiting** and the Status of **Start Point** will change to **Complete**. Finally at the 10 minute mark a message that Event 1 of **End Point** has executed will be displayed. At this point, the integrator will stop and the Status of **End Point** will change to **Complete**.

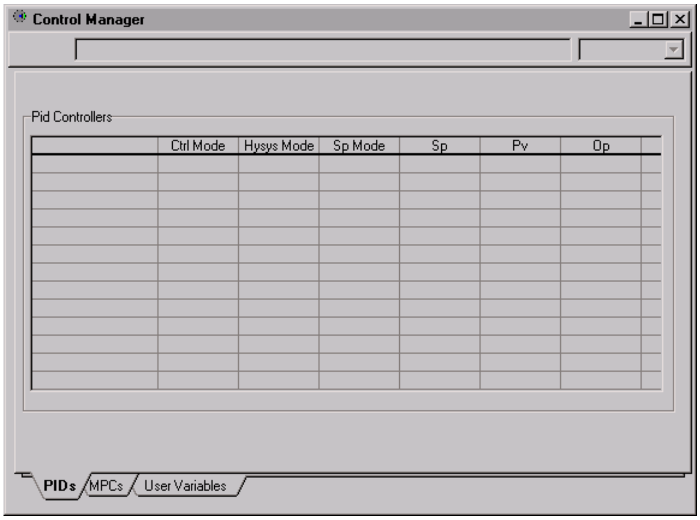
Figure 2.68



2.5 Control Manager

The Control Manager is a summary of the PID Controllers and MPC Controllers contained within the current simulation. There are three tabs available within this view: PID's, MPC's and User Variables.

Figure 2.69



PID Tab

This tab provides a summary of the PID Controllers within the current simulation. There are three modes displayed:

- Controller mode - allows you to set the controller to automatic, manual or off. Refer to section **Section 10.1.3 - Parameters Tab** for more information.
- HYSYS mode - allows you toggle the HYSYS mode between Internal and External.
- Sp mode - will display whether the SP is set to local or remote. Refer to section **Section 10.1.3 - Parameters Tab** for more information.

There are also three key variables displayed: set point (Sp), process variable (Pv), and operating target object (Op).

MPC Tab

This tab provides a summary of the MPC Controllers within the current simulation. There are two modes are displayed:

- Controller mode - allows you to set the controller to automatic, manual or off. Refer to section **Section 10.3.3 - Parameters Tab** for more information.
- HYSYS mode - allows you toggle the HYSYS mode between Internal and External.

There are also three key variables displayed: set point (Sp), process variable (Pv), and operating target object (Op).

User Variable Tab

The User Variables page allows you to create and implement variables in the HYSYS simulation case. Refer to **Chapter 5 - User Variables** in the Customization Guide for more information.

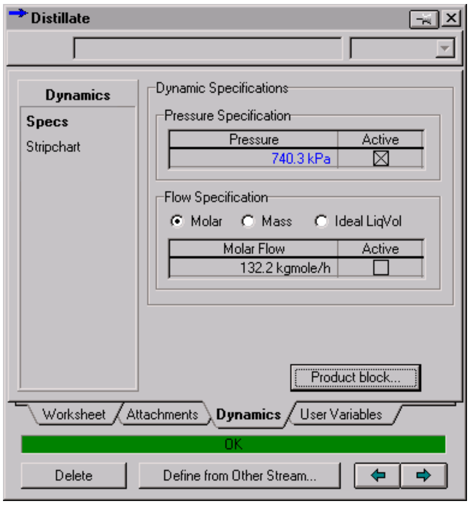
3 Streams

3.1 Material Stream View

3.1.1 Dynamics Tab

The **Dynamics** tab in the stream property view becomes important in **Dynamic** mode. The **Dynamics** tab displays the Pressure-Flow specifications for the material stream as well as users can display the stripchart for a set of variables. The view for the Specs page on the **Dynamics** tab is shown in Figure 3.1.

Figure 3.1



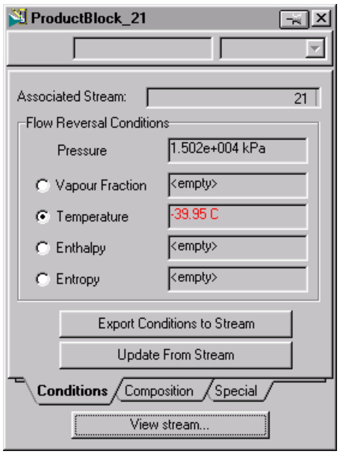
Specs Page

As indicated, a pressure and a flow specification can be made for the stream. If the **Active** box is checked for a specification, the value of the specification becomes *blue* and can be modified by the user. If the active box is not checked, the value is shown in *black* and is calculated by HYSYS.

Feed and Product Blocks

A stream which has only one unit operation attached to it is called a flowsheet boundary stream. If a *material* stream is a flowsheet boundary stream, a ***Feeder block*** or ***Product block*** button is displayed in 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 view displays either a ***Feeder block*** button or a ***Product block*** button. A **Product Block** view of the stream **Distillate** is shown in **Figure 3.2**:

Figure 3.2



The **Feeder Block** view may also be displayed by pressing the left-pointing arrow button (***View Upstream Operation*** button) on the Main stream view. Similarly, the **Product Block** view may be displayed by pressing the right-pointing arrow button (***View Downstream Operation*** button). Default stream conditions are shown in *red*.

The ***Product block*** button opens a view which displays flow reversal conditions of the material stream which the user 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 *Feed block* is used to restore desired feed conditions and compositions if the feed stream reverses and then becomes feed again.

Both *Feed* and *Product Blocks* have similar views. The user is able to specify the stream conditions as follows:

Required Feed and Product Block Specifications	
Conditions Tab	Specify one of: • Temperature • Vapour Fraction • Entropy • Enthalpy
Composition Tab	Specify Stream Composition

Since the pressure of the stream remains the same after the product stream flow reverses, this value does not need to be specified. With this information, the stream is able to perform flash calculations on the other stream properties.

The *Feed* and *Product block* have three buttons used to manipulate the direction of stream conditions between the material stream and the block:

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.

To ignore the *Feed* or *Product block* operations, select the **Ignored** check-box on the **Special** tab.

Stripchart Page

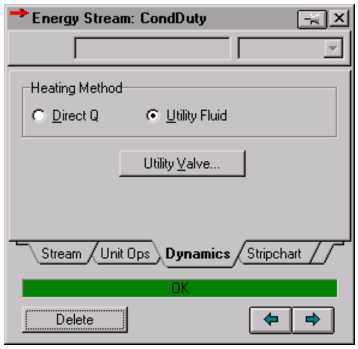
The **Stripchart** page allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the ***Create Stripchart*** button to display the stripchart.

3.2 Energy Stream View

3.2.1 Dynamics Tab

In **Dynamic** mode, two different heating methods can be chosen for an energy stream. When the **Direct Q** Duty Source is chosen, a duty value can be specified by the user. When the **Utility Fluid** radio button is chosen, the duty is calculated from specified properties of a utility fluid. The **Dynamics** tab of stream **Q-100** is shown in Figure 3.3:

Figure 3.3



The ***Utility Valve*** button opens the **Flow Control Valve** (**FCV**) view for the energy stream. A detailed description of the **Direct Q** and **Utility Fluid Heating** methods are given in Section 5.2 - Valve.

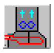
3.2.2 Stripchart Tab

The **Stripchart** tab allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the ***Create Stripchart*** button to display the stripchart.

4 Heat Transfer Equipment

4.1 Air Cooler

The dynamics **Air Cooler** unit operation exchanges heat with an ideal air mixture. In dynamics, the air flow must be calculated using the fan rating information. The steady state and dynamic air cooler operations share the same rating information.



Air Cooler Button

The dynamic **Air Cooler** tube, unlike the steady state **Air Cooler**, is capable of storing inventory like other dynamic unit operations. The direction of flow of material through the Air Cooler is governed by the pressures of the surrounding unit operations.

To install the **Air Cooler** operation, press **F12** and choose **Air Cooler** from the **UnitOps** view or select the ***Air Cooler*** button in the Object Palette.

To ignore the **Air Cooler**, select the **Ignore** check box. HYSYS will completely disregard the operation (and will not calculate the outlet stream) until you restore it to an active state by clearing the check box.

4.1.1 Theory

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:

$$M_{process}(H_{in} - H_{out})_{process} - M_{air}(H_{in} - H_{out})_{air} = \rho \frac{d(VH_{out})_{process}}{dt} \tag{4.1}$$

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*

The **Air Cooler** duty, Q, can be defined in terms of the overall heat transfer coefficient, the area available for heat exchange and the log mean temperature difference:

$$Q = -UA\Delta T_{LM}F_t \qquad (4.2)$$

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.

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:

$$flow = \sqrt{density} \times k\sqrt{P_1 - P_2} \qquad (4.3)$$

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 **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:

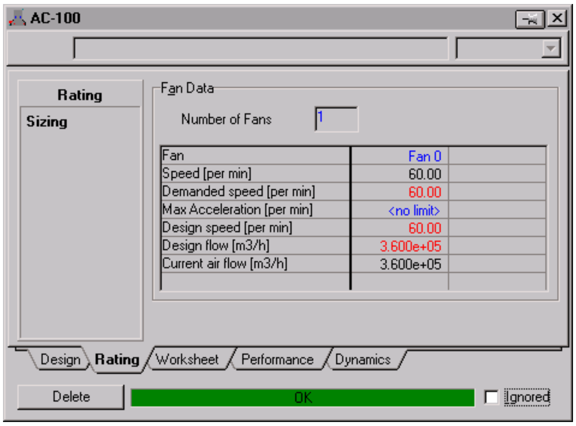
Dynamic Specifications	Description
Overall UA	This is the product of the **Overall Heat Transfer Coefficient** and the total area available for heat transfer. The Overall UA must be specified in **Dynamic** mode. You can specify the value of UA in the **Parameters** page of the **Design** tab.
Fan Rating Information	You must specify the following information in 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 in the **Specs** page of the **Dynamics** tab.

4.1.2 Rating Tab

The **Rating** tab contains one page, the **Sizing** page.

Sizing Page

Figure 4.1



In the **Sizing** page, the following fan rating information is displayed for the **Air Cooler** operation:

Fan Data	Description
Number of Fans	Specify the number of fans you want in the air cooler.
Speed	This is the actual speed of the fan in rpm (rotations per minute).
Demanded Speed	This is the desired speed of the fan. In **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	This is the rate at which the actual speed moves to the demanded speed.
Design Speed	This is the reference Air Cooler fan speed. It is used in the calculation of the actual air flow through the cooler.
Design Flow	This is the reference Air Cooler air flow. It is used in the calculation of the actual air flow through the cooler.
Current Air Flow	This may be calculated or specified by the user. If the air flow is specified no other fan rating information needs to be specified.

The air flow through the fan is calculated using a linear relation:

$$Fan\ Air\ Flow = \frac{Speed}{Design\ Speed} \times Design\ Flow \tag{4.4}$$

Unlike the steady state **Air Cooler**, the actual speed of the fan will not always equal the demanded speed. The actual fan speed after each integration time step is calculated as follows:

$$\begin{aligned} &Actual\ Speed = (Max\ Acceleration)\Delta t + Actual\ Speed_o \\ &\text{until } Actual\ Speed = Demanded\ Speed \end{aligned} \tag{4.5}$$

Each fan in the air cooler contributes to the air flow through the cooler. The total air flow is calculated:

$$Total\ Air\ Flow = \sum Fan\ Air\ Flow \tag{4.6}$$

Nozzles Page

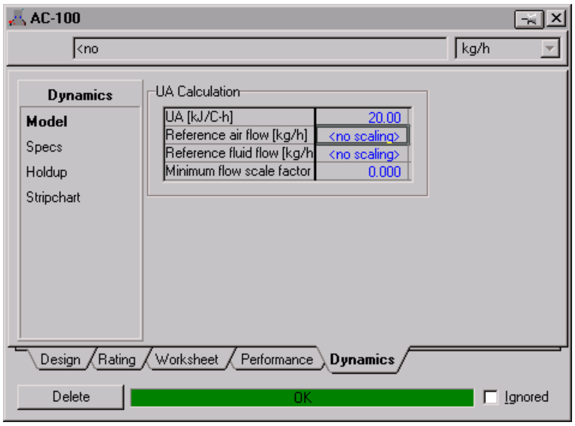
The **Nozzles** page contains information regarding the elevation and diameter of the nozzles. The Plant+ license is required to use the Nozzle features found on this page. Refer to **Section 1.5 - Plant+** for more information.

4.1.3 Dynamics Tab

The **Dynamics** tab contains four pages: the **Model**, **Specs, Holdup** and **Stripchart** page.

Model Page

Figure 4.2



The **Model** page allows you to define how UA is defined in Dynamic mode. The value of *UA* is calculated by:

$$UA_{dynamic} = F \times UA_{steadystate} \tag{4.7}$$

where: $UA_{steadystate}$ *= the UA value entered on the* ***Parameters*** *page of the* ***Design*** *tab.*

$$F = \frac{2 \times f1 \times f2}{(f1 + f2)} \quad \textit{the flow scale factor} \tag{4.8}$$

$$f1 = (mass\ flowrate\ /\ reference\ flowrate)\text{^}0.8\ for\ air. \tag{4.9}$$

$$f2 = (mass\ flowrate\ /\ reference\ flowrate)\text{^}0.8\ for\ fluid. \tag{4.10}$$

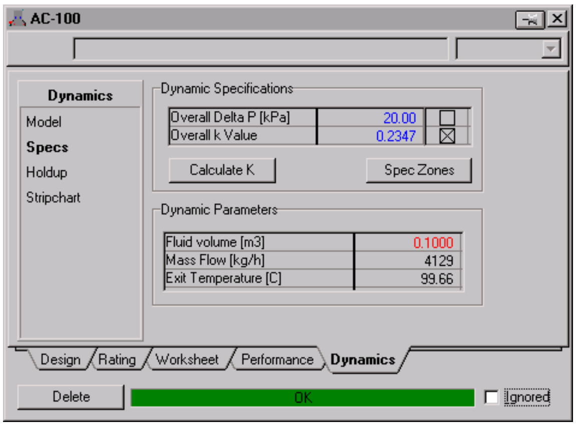
The page consists of one group: **UA Calculation**. It contains four fields:

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	This is the reference flowrate for air. It is used to calculate the value of *f*1 as shown in **Equation (4.9)**.
Reference fluid flow	This is the reference flowrate for air. It is used to calculate the value of *f*1 as shown in **Equation (4.10)**.
Minimum flow scale factor	This is the minimum scale factor used. If the value calculated by **Equation (4.8)** is smaller than this value, this value will be used.

Specs Page

The **Specs** page contains information regarding the calculation of pressure drop across the **Air Cooler**:

Figure 4.3



You can specify how the pressure drop across the **Air Cooler** is calculated in the **Dynamic Specifications** group box

*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.*

Dynamic Specification	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 check box.
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 pressing the ***Calculate k*** button. Ensure that there is a non zero pressure drop across the **Air Cooler** before the ***Calculate k*** button is pressed. To use the k-value as a dynamic specification, select the corresponding check box.
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.

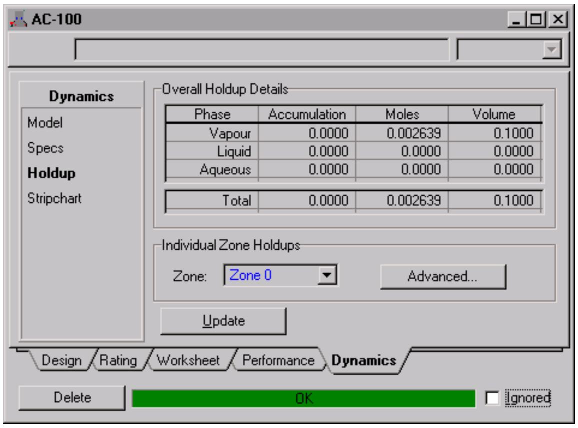
The **Dynamic Parameters** section contains information about the holdup of the **Air Cooler**:

Dynamic Parameter	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	This is the exit temperature of the process stream.

Holdup Page

The **Holdup** page contains information regarding the holdup's properties, composition, and amount.

Figure 4.4



For each phase contained within the volume space of the unit operation, the following is specified:

Holdup Details	Description
Holdup Volume	The holdup volume can be specified in this cell.
Accumulation	The accumulation refers to the rate of change of material in the holdup for each phase.
Moles	These cells indicate the amount of material in the holdup for each phase.
Volume	These cells indicate the holdup volume of each phase.

Click the Advanced button to accesses a view that provides more detailed information about the holdup of that unit operation. Refer to Section 1.2.7 - Advanced Holdup Properties for more information.

Stripchart Page

The **Stripchart** page allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the **Create Stripchart** button to display the stripchart.

4.1.4 Air Cooler Example

An **Air Cooler** will be used to cool to a liquid process stream.

1. Create the following Fluid Package:

Property Package	Components
NRTL	Ethanol, 1-Pentanol

2. Specify stream **Feed** with the following properties:

Name	Feed
Temperature [F]	100.0000
Pressure [psi]	16.0000
Molar Flow [lbmole/hr]	2200.0000
Comp Mole Frac [Ethanol]	0.5
Comp Mole Frac [1-Pentanol]	0.5

3. Install the **Air Cooler** with the following connections and parameters as shown below:

UNIT OPERATION TYPE	Air Cooler	
Tab [Page]	**Input Area**	**Entry**
Design [Connections]	Name	AC-100
	Feed	Feed
	Product	Product
Design [Parameters]	Delta P	1.2 psi
	Overall UA	1.52×10^5 Btu/F-hr
	Configuration	1 tube row, 1 pass
	Air Intake Temperature	77 F

4. On the **Sizing** page of the **Ratings** tab, enter the fan rating and speed information. Using the following specifications, HYSYS is able to calculate the actual volumetric flow of air:

Tab [Page]	Input Area	Entry
Rating [Sizing]	Number of Fans	1
	Demanded Speed	60 rpm
	Max Acceleration	0.5 rpm^2
	Design Speed	60 rpm
	Design Flow	5.434×10^7 barrel/day

Dynamics Mode Button

5. Switch to **Dynamics** mode.
6. On the **Specs** page of the **Dynamics** tab, confirm that **Overall Delta P** of the Air Cooler is specified as 1.2 psi. Also confirm that the mass flow rate in the **Dynamic Parameters** section is 1.476 x 10^5 lb/hr (2200 lbmole/hr). These parameters are used by HYSYS to determine the k-value for the Air Cooler.
7. Calculate the k-value by pressing the ***Calculate k*** button. Deactivate the **Overall Delta P** specification and activate the **Overall K value** specification.

Figure 4.5

Overall Delta P [psi]	1.200	☐
Overall k Value	0.2341	☒

8. On the **PF Specs** page of the **Worksheet** tab, deactivate the *Flow Spec* of stream **Feed** and activate the *Pressure Spec* of stream **Product**. Specify the pressure of stream **Product** to be 14.8 psi.
9. At this point, the **Air Cooler** unit operation is fully specified. Run the Integrator until all the variables line out. The **Product** stream conditions are displayed in the **Conditions** page of the **Worksheet** tab.

Figure 4.6

Name	Feed	Product
Vapour	0.0000	0.0000
Temperature [F]	100.0	84.60
Pressure [psia]	16.00	14.80
Molar Flow [lbmole/hr]	2200	2200
Mass Flow [lb/hr]	1.476e+05	1.476e+05
LiqVol Flow [barrel/day]	1.246e+04	1.246e+04
Molar Enthalpy [Btu/lbmole	-1.348e+05	-1.355e+05
Molar Entropy [Btu/lbmole-F	14.31	14.31
Heat Flow [Btu/hr]	-2.966e+08	-2.982e+08

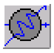
10. The **Air Cooler** unit operation parameters are shown in the **Results** page of the **Performance** tab:

Figure 4.7

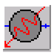
Working Fluid Duty [Btu/hr]	-1.538e+06
Correction Factor	0.8917
UA [Btu/F-hr]	1.520e+05
LMTD [F]	11.35
Feed T [F]	100.0
Product T [F]	84.60
Air Inlet T [F]	77.00
Air Outlet T [F]	83.83
Total vol. Air Flow [barrel/day]	5.434e+07
Total Mass Air Flow [lb/hr]	9.300e+05

4.2 Cooler/Heater

The dynamic **Cooler** and **Heater** operations are one-sided heat exchangers. The inlet stream is cooled (or heated) to required exit conditions and the energy stream absorbs (or provides) the energy between the two streams. These operations are useful when you are only interested in how much energy is required to cool or heat a process stream with a utility, but are not interested in the conditions of the utility itself.



Cooler Button



Heater Button

To install the **Cooler/Heater** operation, press **F12** and choose **Cooler** or **Heater** from the **UnitOps** view or select the **Cooler** or **Heater** button in the Object Palette.

To ignore the **Cooler/Heater**, select the **Ignore** check box. HYSYS will completely disregard the operation (and will not calculate the outlet stream) until you restore it to an active state by clearing the check box.

4.2.1 Theory

Heat Transfer

The **Cooler** and **Heater** use the same basic equation. 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's** process side holdup:

$$M(H_{in} - H_{out}) - Q_{cooler} = \rho\frac{d(VH_{out})}{dt} \tag{4.11}$$

For a **Heater**, the enthalpy or heat flow of the energy stream is added to the **Heater's** process side holdup:

$$M(H_{in} - H_{out}) + Q_{heater} = \rho\frac{d(VH_{out})}{dt} \tag{4.12}$$

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.
- 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. This relation is similar to the general valve equation:

$$flow = \sqrt{density} \times k\sqrt{P_1 - P_2} \tag{4.13}$$

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:

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 in the **Specs** page of the **Dynamics** tab.
Pressure Drop	Either specify an Overall Delta P or an Overall K-value for the Air Cooler. Specify the Pressure Drop calculation in the **Dynamic Specifications** group in the **Specs** page of the **Dynamics** tab.

4.2.2 Rating Tab

Nozzle Page

The **Nozzles** page contains information regarding the elevation and diameter of the nozzles. The Plant+ license is required to use the Nozzle features found on this page. Refer to **Section 1.5 - Plant+** for more information.

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 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 specified values:

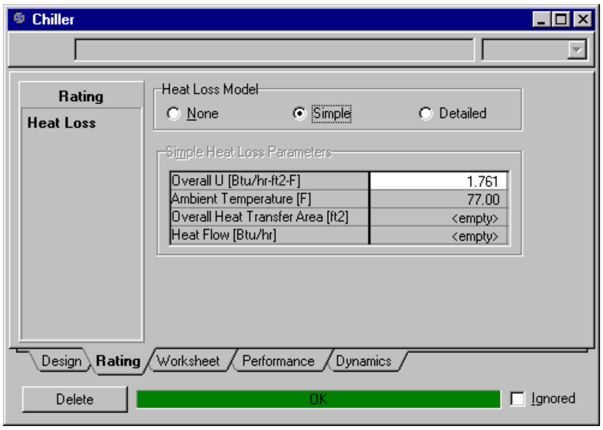
- Overall U value
- Ambient Temperature

The heat transfer area, A, and the fluid temperature, T_f, are calculated by HYSYS.Plant. The heat loss is calculated using:

$$Q = UA(T_f - T_{amb}) \tag{4.14}$$

For a separator the parameters available for Simple model are shown in **Figure 4.8**:

Figure 4.8



The simple heat loss parameters are:

- The Overall Heat Transfer Coefficient
- Ambient Temperature
- Overall Heat Transfer Area
- Heat Flow

The Heat Flow is calculated as follows:

$$Heat\ Flow = UA(T_{Amb} - T) \tag{4.15}$$

where: U = *the overall heat transfer coefficient*

A = *the heat transfer area*

T_{Amb} = *the 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, may be modified from their default values (shown in red).

Detailed Model

The Detailed model allows you to specify more detailed heat transfer parameters. The Plant+ license is required to use the Detailed Heat Loss model found on this page. Refer to Section 1.5 - Plant+ for more information.

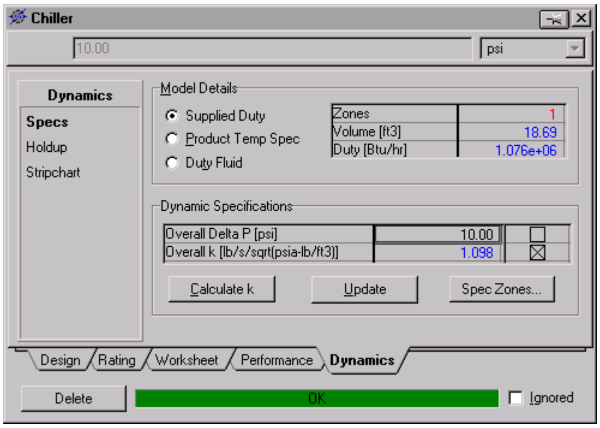
4.2.3 Dynamics Tab

The **Dynamics** tab contains four pages: the **Specs**, **Duty Fluid**, **Holdup** and **Stripchart** page.

Specs Page

The **Specs** page contains information regarding the calculation of pressure drop across the **Cooler** or **Heater**:

Figure 4.9



Zone Information

HYSYS has the ability to partition the **COOLER** and **HEATER** into discrete sections called zones. By dividing this unit operation into zones, the user can make different heat transfer specifications for individual zones and therefore more accurately model the physical process. However, specifying the **Cooler/Heater** with one zone provides optimal speed conditions and is usually sufficient in modeling accurate exit stream conditions. It is therefore not suggested to specify the **COOLER** or **HEATER** with more than one zone.

Model Details

The **Model Details** group box must be completed before the simulation case will solve. The number of zones and the volume of a **Cooler/ Heater** can be specified in the **Model Details** group. HYSYS can calculate the duty applied to the holdup fluid using three different methods:

Model	Description
Supplied Duty	If the **Supplied Duty** model is selected, 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 the **Product Temp Spec** model is selected, you must specify the desired exit temperature. HYSYS back calculates the required duty to achieve the specified desired temperature. This method may not run as fast as the **Supplied Duty** model.
Duty Fluid	If the **Duty Fluid** model is selected, you can model a simple utility fluid to heat or cool your process stream. The following parameters must be specified for the utility fluid in the **Duty Fluid** page of the **Dynamics** tab: • **Mass Flow** • **Holdup Mass** • **Mass Cp** • **Inlet temperature** • **Average UA**

Dynamic Specifications

You can specify how the pressure drop is calculated across the **Cooler/ Heater** unit operation in the **Dynamic Specifications** section.

*The **Cooler/Heater** 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.*

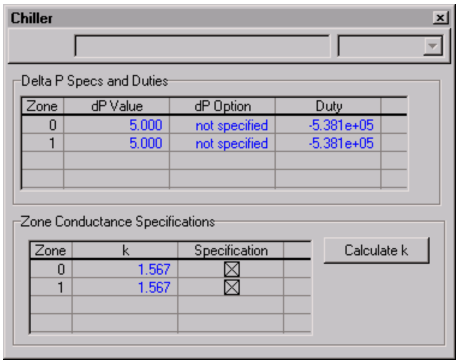
Dynamic Specification	Description
Overall Delta P	A set pressure drop is assumed across the **Cooler/ Heater** operation with this specification. The flow and the pressure of either the inlet or exit stream must be specified or calculated from other unit operations in the flowsheet. The flow through the valve is not dependent on the pressure drop across the **Cooler/Heater**. To use the overall delta P as a dynamic specification, select the corresponding box in the **Dynamic specifications** section

Dynamic Specification	Description
Overall k Value	The k-value defines the relationship between the flow through **Cooler/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/Heater** with a k-value by pressing the ***Calculate k*** button. Ensure that there is a non zero pressure drop across the **Cooler/Heater** before the ***Calculate k*** button is pressed. To use the k-value as a dynamic specification, select the corresponding box in the **Dynamic specifications** section.

Zone Dynamic Specifications

If the **Cooler/Heater** operation is specified with multiple zones, you can select the ***Spec Zones*** button to define dynamic specifications for each zone.

Figure 4.10



In the **Delta P Specs and Duties** group box, you can specify the following parameters:

Dynamic Specification	Description
dP Value	The fixed pressure drop value is specified in this field.

Dynamic Specification	Description
dP Option	This field allows you to either specify or calculate the pressure drop across the **Cooler/Heater**. Specify the dP Option with one of the following options: • user specified - The pressure drop across the zone is specified by the user 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 box.
Duty	A fixed duty can be specified across each zone in the **Cooler/Heater** unit operation.

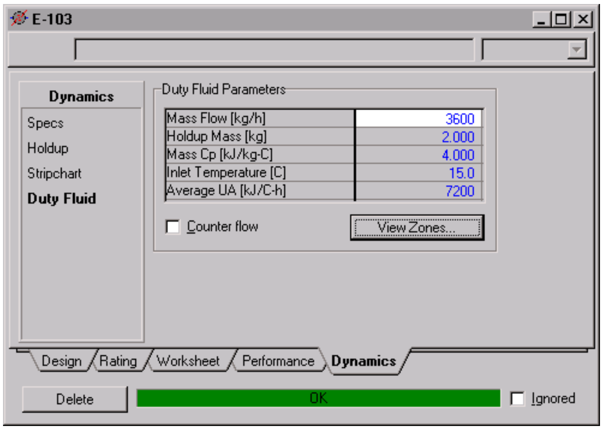
In the **Zone Conductance Specifications** group box, you can specify the following parameters:

Dynamic Specification	Description
k	The k-value for individual zones may be specified in this field. You can either specify the k-value or have it calculated by pressing 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** model is specified in the **Specs** page.

Figure 4.11



This page allows you to enter the following parameters to define your duty fluid:

- Mass Flow
- Holdup mass
- Mass Cp
- Inlet Temperature
- Average UA

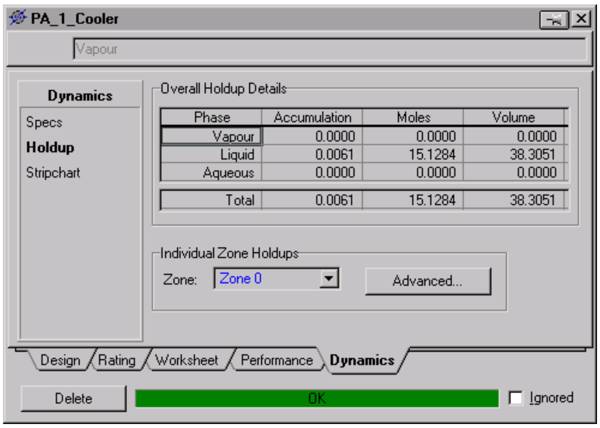
The **Counter Flow** check box allows you to specify the direction of flow for your the duty fluid. When the check box is active you are using a counter flow.

The **View Zones** button displays the duty fluid parameters for each of the zones specified on the **Specs** page.

Holdup Page

The **Holdup** page contains information regarding the **Cooler/Heater** holdup properties, composition, and amount.

Figure 4.12



The **Overall Holdup Details** section contains information regarding the **Cooler/Heater** holdup parameters. For each phase contained within the volume space of the unit operation, the following is specified:

Holdup Details	Description
Accumulation	The accumulation refers to the rate of change of material in the holdup for each phase.
Moles	These cells indicate the amount of material in the holdup for each phase.
Volume	These cells indicate the holdup volume of each phase.

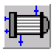
The **Individual Zone Holdups** section contains detailed holdup properties for each holdup in the **Cooler/Heater**. In order to view the advanced properties for individual holdups, you must first choose the individual zone in the **Zone** drop down list and then press the **Advanced** button.

Click the Advanced button to accesses a view that provides more detailed information about the holdup of that unit operation. Refer to Section 1.2.7 - Advanced Holdup Properties for more information.

Stripchart Page

The **Stripchart** page allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the ***Create Stripchart*** button to display the stripchart.

4.3 Heat Exchanger



Heat Exchanger Button

The dynamic Heat Exchanger performs energy and material balances in a shell-and-tube heat exchanger. Like the steady state heat exchanger, the dynamic heat exchanger can be specified with either detailed rating information or simple design specifications. The steady state and dynamic heat exchanger operations share the same rating information.

Unlike the steady state heat exchanger, the shell and tube of the dynamic heat exchanger is capable of storing inventory like other dynamic vessel operations. The direction of flow of material through the dynamic heat exchanger is governed by the pressures of the surrounding unit operations.

Some of the key features included in the dynamic **Heat Exchanger** operation include:

- 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 modelled.
- 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.

To install the Heat Exchanger operation, press **F12** and choose **Heat Exchanger** from the **UnitOps** view or select the **Heat Exchanger** button in the Object Palette.

To ignore the Heat Exchanger, select the **Ignore** check box. HYSYS will completely disregard the operation (and will not calculate the outlet stream) until you restore it to an active state by clearing the check box.

4.3.1 Theory

Heat Transfer

The Heat Exchanger calculations are based on energy balances for the hot and cold fluids. The following general relation applies to the shell side of the Basic model Heat Exchanger.

$$M_{shell}(H_{in}-H_{out})_{shell}-Q_{loss}+Q = \rho\frac{d(VH_{out})_{shell}}{dt} \tag{4.16}$$

For the tube side:

$$M_{tube}(H_{in}-H_{out})_{tube}-Q = \rho\frac{d(VH_{out})_{tube}}{dt} \tag{4.17}$$

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*

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, see **Section 1.2.4 - Heat Loss Model** in this manual.

The total heat transferred between the tube and shell sides (**Heat Exchanger** duty) may be defined in terms of the overall heat transfer coefficient, the area available for heat exchange and the log mean temperature difference:

$$Q = UA\Delta T_{LM}F_t \tag{4.18}$$

where: U = *Overall heat transfer coefficient*

A = *Surface area available for heat transfer*

ΔT_{LM} = *Log mean temperature difference (LMTD)*

F_t = *LMTD correction factor*

> Note that the heat transfer coefficient and the surface area are often combined for convenience into a single variable referred to as UA.

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 according to 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:

$$f = \sqrt{density} \times k\sqrt{P_1 - P_2} \quad (4.19)$$

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 is a list of the minimum specifications required for the **Heat Exchanger** unit operation to solve.

The **Basic** Heat Exchanger model requires the following specifications:

Dynamic Specifications	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 in the **Specs** page of the **Dynamics** tab. You can also specify the Overall Delta P values for the shell and tube sides in the **Sizing** page of the **Rating** tab.

The **Detailed** Heat Exchanger model requires the following specifications:

Dynamic Specifications	Description
Sizing Data	The tube and shell sides of the **Heat Exchanger** must be completely specified in 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.

Dynamic Specifications	Description
Overall UA	Either specify an Overall UA or have it calculated from the Shell and Tube geometry. Specify the U calculation method in the **Parameters** page of the **Rating** tab. The U calculation method may also be specified in 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 in the **Parameters** page of the **Rating** tab. You can also specify the Pressure Drop calculation method in the **Pressure Flow Specifications** group in the **Specs** page of the **Dynamics** tab.

4.3.2 Rating Tab

The **Rating** tab contains four pages: the **Sizing**, **Parameters**, **Nozzles** and **Heat Loss** page.

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.

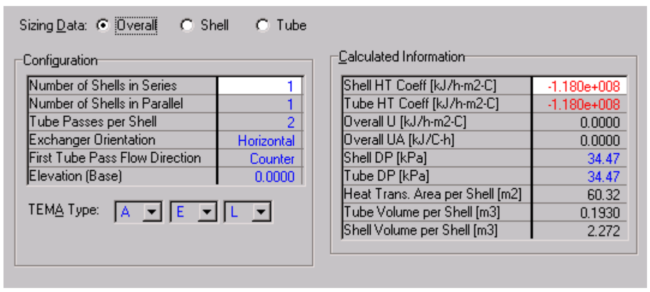
The information is organized by three radio buttons: **Overall**, **Shell**, and **Tube**.

Overall

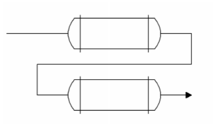
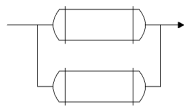
Tube flow direction can be defined as either ***Counter*** *or* ***Co-Current*** *for all Heat Exchanger calculation models.*

If the **Overall** radio button is selected, the overall Heat Exchanger geometry is displayed:

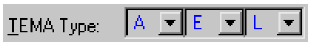
Figure 4.13



In the **Configuration** section, you can specify whether multiple shells will be used in the heat exchanger design. The following fields are displayed and can be modified in the **Configuration** section.

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:  Note: Currently, multiple shells in parallel are not supported in HYSYS.
Tube Passes per Shell	The number of tube passes per shell. Usually 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 a checkbox: 
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 may 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 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 is used in the **Dynamics** Heat Exchanger Model in the calculation of liquid level in the shell.



TEMA Type Drop down Lists

The shape of Heat Exchanger may 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. 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).

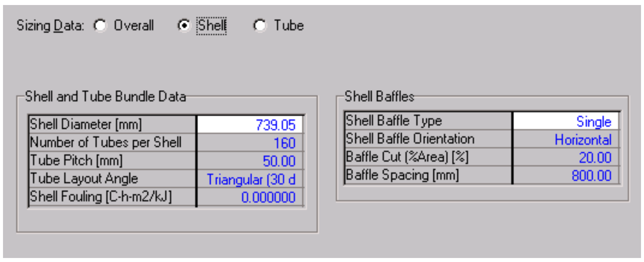
In the **Calculated Information** group, the following heat exchanger parameters are listed:

- 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

If the **Shell** radio button is selected, the shell configuration and the baffle arrangement in each shell can be specified.

Figure 4.14



In the **Shell and Tube Bundle Data** section, you can specify whether multiple shells will be used in the heat exchanger design. The following fields are displayed and can be modified in this section.

Field	Description
Shell Diameter	The diameter of the shell(s) is shown in this field.
Number of Tubes per Shell	This is the number of tube holes on the tube sheet.
Tube Pitch	This is the shortest distance between the centres of two adjacent tubes.
Tube Layout Angle	In HYSYS, the tubes in the a single shell may 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.

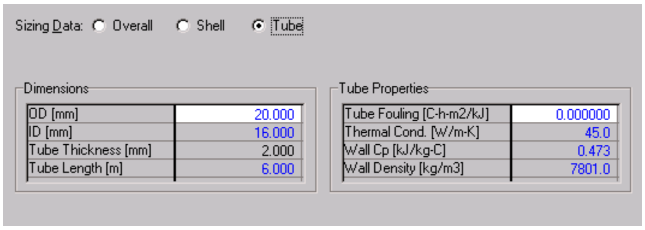
The following fields are displayed and can be modified in the **Shell Baffles** section:

Field	Description
Shell Baffle Type	You can choose 4 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 the baffle occupies relative to the cross sectional area of the shell.
Baffle Spacing	You can specify the space between each baffle.

Tube

If the **Tube** radio button is selected, information about the tube geometry in each shell can be specified:

Figure 4.15



In the **Dimensions** group, the following tube geometric parameters may be specified:

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	This is the heat transfer length of one tube in a single heat exchanger shell. Note that 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

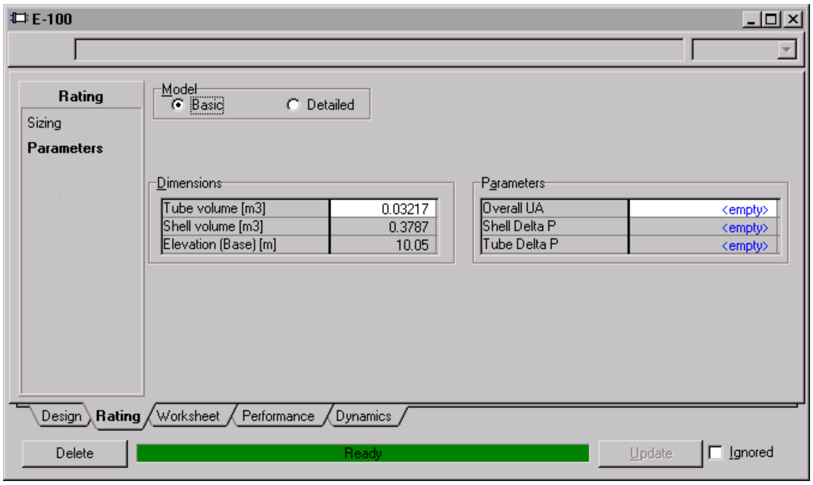
Parameters Page

The **Parameters** page of the **Rating** tab is used to define rating parameters for the **Dynamic Rating** model or the **Dynamics** Heat Exchanger 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's 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

Selecting the **Basic** model radio button on the **Parameters** page in **Dynamics** mode will bring up the following view:

Figure 4.16



The **Dimensions** section 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 may be specified but does not impact the steady state solver.

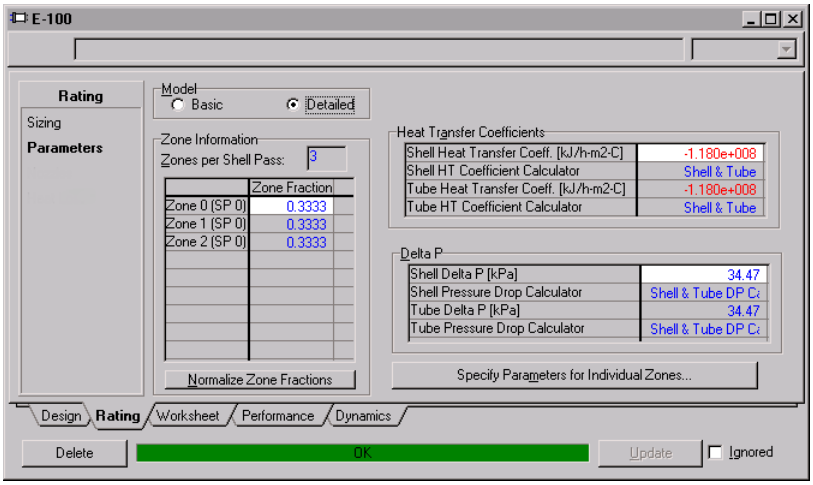
The **Parameters** section includes the following Heat Exchanger parameters. All but the correction factor, F, may be modified:

Field	Description
Overall UA	This is 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 may 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 may be specified here. If you do not specify the DP values, HYSYS will calculate them from the attached stream pressures.

Detailed Model

The **Detailed** model option allows you to specify the **Zone Information**, **Heat Transfer Coefficient**, and **DeltaP** details. Selecting the **Detailed** model radio button on the **Parameters** page will bring up the following view shown in Figure 4.17.

Figure 4.17



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** section you can specify:

Field	Description
Zones per Shell Pass	Enter the number of zones you would like 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 zone fraction is the fraction of space the zone occupies relative to the total shell volume. HYSYS automatically sets each zone to have the same volume. You may modify the zone fractions to occupy a larger or smaller proportion of the total volume. Press the ***Normalize Zone Fractions*** button in order to adjust the sum of fractions to equal one.

Heat Transfer Coefficients

The **Heat Transfer Coefficients** section 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 in the **Parameters** page and the stream conditions.
- The user 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:

$$h_i = \frac{0.023 D_i}{k_m}\left(\frac{D_i G_i}{\mu_i}\right)^{0.8}\left(\frac{C_{p,i}\mu_i}{k_m}\right)^{1/3}\left(\frac{\mu_i}{\mu_{i,w}}\right)^{0.14} \tag{4.20}$$

where: G_i = *Mass velocity of the fluid in the tubes (Velocity* Density)*

m_i = *Viscosity of the fluid in the tube*

$m_{i,w}$ = *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 (4.21)**.

$$U = \frac{1}{\left[\frac{1}{h_o} + r_o + r_w + \frac{D_o}{D_i}\left(r_i + \frac{1}{h_i}\right)\right]} \tag{4.21}$$

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 following information is provided in the Heat Transfer coefficients group:

Field	Description
Shell/Tube Heat Transfer Coefficient	The local Heat Transfer Coefficients, h_o and h_i, may 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 the user.

Delta P

The **Delta P** section 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 following information is provided in the **Delta P** section:

Field	Description
Shell/Tube Delta P	The pressure drops 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 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 the user. • **Non specified** - This option is only applicable in Dynamics mode. Pressure drop across the heat exchanger is calculated from a pressure flow relation.

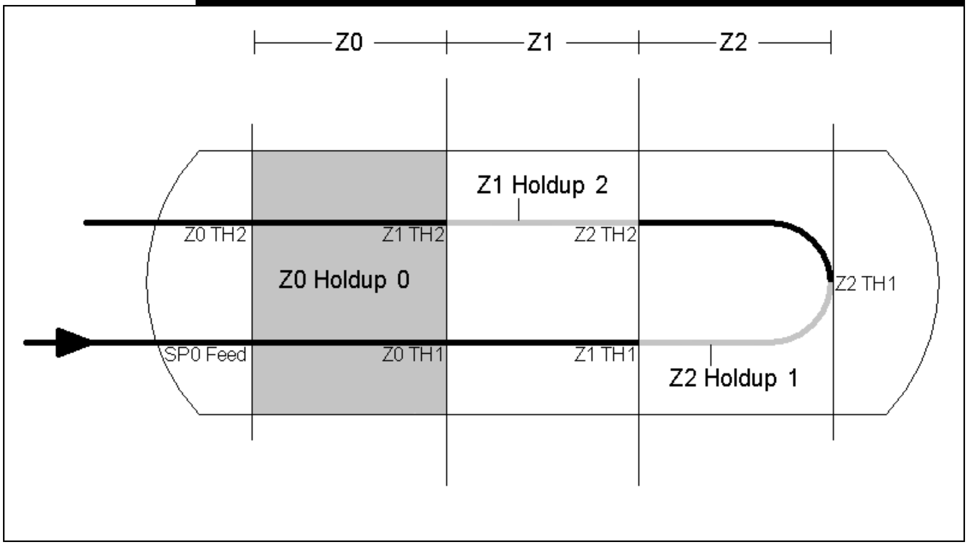
Detailed Heat Model Properties

By pressing the ***Specify Parameters for Individual Zones*** button, a view appears detailing heat transfer parameters and holdup conditions for each zone appears. 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 labelled: Z0, Z1, and Z2.
Holdup	HYSYS represents the holdup within each zone with the letter “H”. Holdups are numbered starting from 0. “Holdup 0” will always be the holdup of the shell within the zone. Holdups 1 through n will represent 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 may 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.

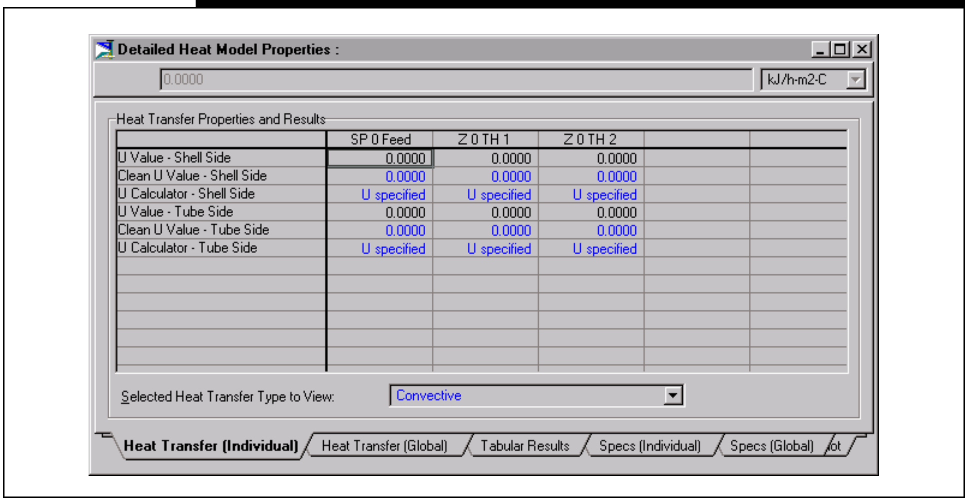
Consider a shell and tube heat exchanger with 3 zones, 1 shell pass, and 2 tube passes. The following diagram labels zones, tube locations, and hold-ups within the heat exchanger:

Figure 4.18



Heat Transfer (Individual) Tab

Figure 4.19



Information regarding the heat transfer elements of each tube location in the heat exchanger is displayed on the **Heat Transfer (Individual)** tab. 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 may choose whether **Conductive** or **Convective** elements are to be displayed 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 may be calculated by HYSYS or modified by the user.
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 are displayed. The conductivity may be modified by the user.

Heat Transfer (Global) Tab

This tab displays the heat transfer elements for the entire Heat Exchanger. You may choose whether the overall **Conductive** or **Convective** elements are to be displayed 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 are displayed for each zone.

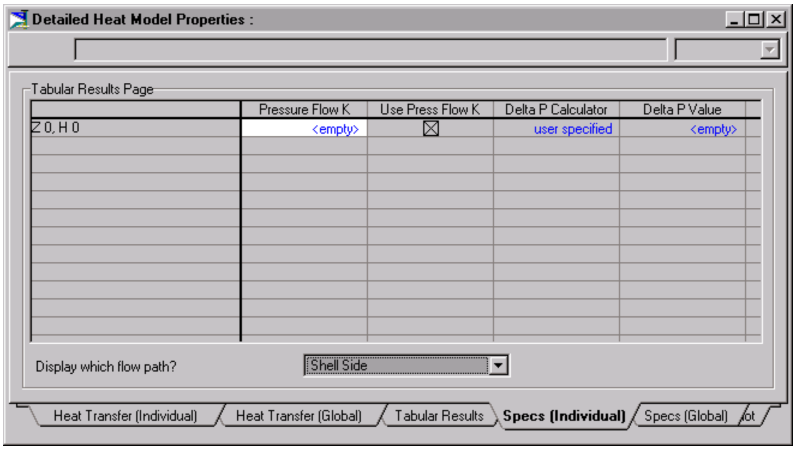
- Temperature
- Pressure
- Vapour Fraction
- Molar Flow
- Enthalpy
- Cumulative UA
- Cumulative Heat Flow

- Length (into heat exchanger)

You may choose whether the flow path is shell or tube side by selecting the appropriate flow path in the **Display which flowpath?** drop-down list.

Specs (Individual) Tab

Figure 4.20



This tab displays the pressure drop specifications for each shell and tube holdup in one spreadsheet. The **Pressure Flow K** and **Use Pressure Flow K** columns are applicable only in Dynamics mode.

You may choose whether the shell or tube side is displayed by selecting the **Display which flowpath?** drop-down list.

Specs (Global) Tab

This 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 **Dynamics** mode.

You may choose whether the shell or tube side is displayed by selecting the **Display which flowpath?** drop-down list.

Plots Tab

The information displayed in this tab is a graphical representation of the parameters provided in 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

The **Nozzles** page contains information regarding the elevation and diameter of the nozzles. The Plant+ license is required to use the Nozzle features found on this page. Refer to **Section 1.5 - Plant+** for more information.

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 will draw material from the liquid holdup. If the product nozzle is located above the liquid level, the exit stream will draw material from the vapor 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 vapor product nozzle does not necessarily produce pure vapor. A 3-phase separator may not produce two distinct liquid phase products from its product nozzles.

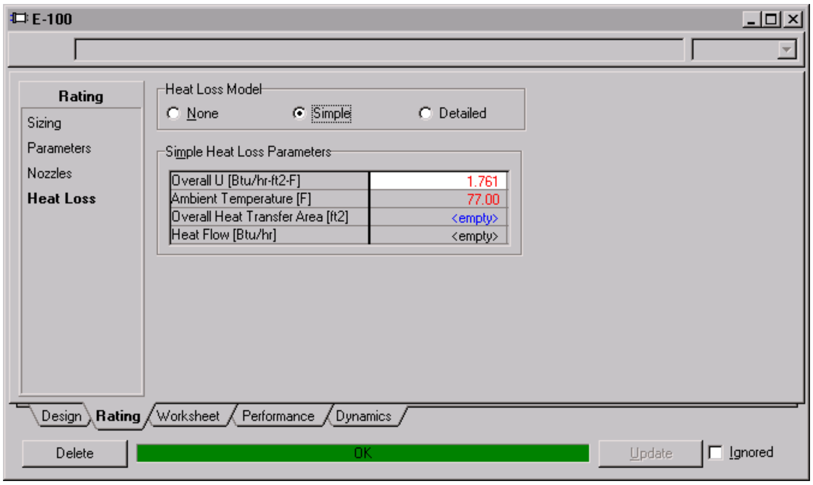
Heat Loss Page

The **Heat Loss** page contains heat loss parameters which characterize the amount of heat lost across the vessel wall.

You can choose either a have no heat loss model, a **Simple** heat loss model or a **Detailed** heat loss model.

Simple Heat Loss Model

Figure 4.21



Selecting the **Simple** radio button displays the following parameters that are used to calculate heat loss.

- Overall U
- Ambient Temperature
- Overall Heat Transfer Area
- Heat Flow

Detailed Heat Loss Model

The Detailed model allows you to specify more detailed heat transfer parameters. The Plant+ license is required to use the Detailed Heat Loss model found on this page. Refer to Section 1.5 - Plant+ for more information.

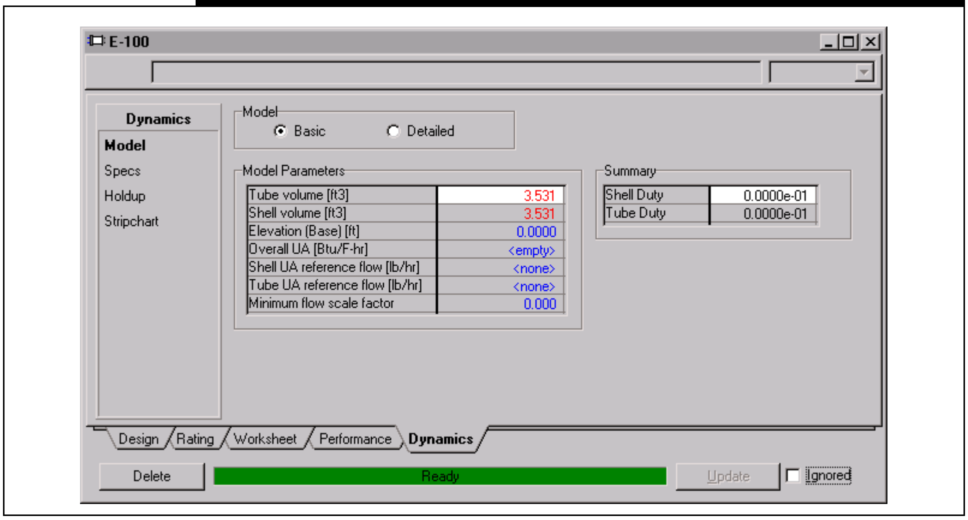
4.3.3 Dynamics Tab

The **Dynamics** tab contains four pages: the **Model**, **Specs**, **Holdup** and **Stripchart**.

Model Page

In the **Model** page, you can specify whether HYSYS will use a **Basic** or **Detailed** model.

Figure 4.22



Basic Model

The **Model Parameters** section contains the following information for the **Heat Exchanger** unit operation:

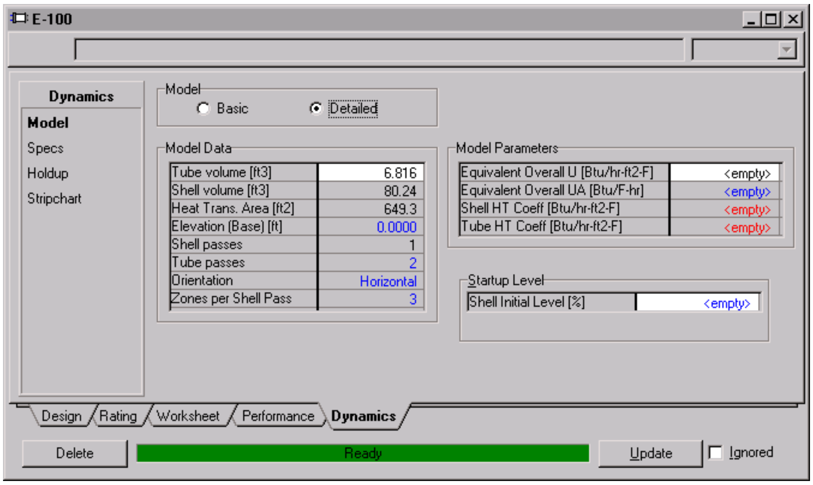
Field	Description
Tube/Shell Volume	The volume of the shell and tube must be specified in the Basic model.
Elevation	The elevation is significant in the calculation of static head around and in the heat exchanger.
Overall UA	This is the product of the Overall Heat Transfer Coefficient and the Total Area available for heat transfer. The heat exchanger duty is proportional to the log mean temperature difference, where UA is the proportionality factor. The UA 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 will be fixed.
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 will still take 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.

The **Summary** section contains information regarding the duty of the heat exchanger shell and tube sides.

Detailed Model

If the **Detailed** radio button displays a summary of the rating information provided in **Rating** tab.

Figure 4.23



The **Model Data** section 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 may 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** section contains the local and overall heat transfer coefficients for the **Heat Exchanger**. Depending on how the **Heat Transfer Coefficient Calculator** is set in the **Parameters** page of the **Rating** tab, the local and overall heat transfer coefficients may either be calculated or specified in the **Model Parameters** section.

HT Coefficient Calculator Setting	Description
Shell & Tube	The overall heat transfer coefficient, U, is calculated using the exchanger rating information.
U Specified	The overall heat transfer coefficient, U, is specified by the user.

The **Startup Level** section is displayed 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 may 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.

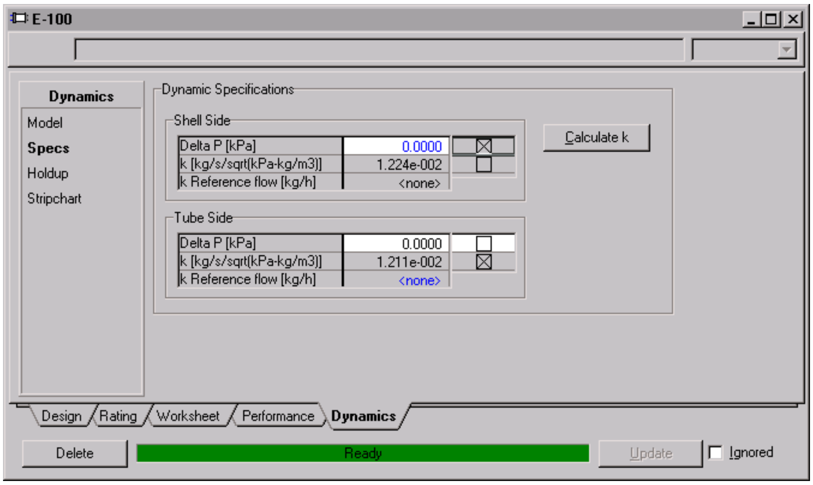
Specs Page

The **Specs** page contains information regarding the calculation of pressure drop across the **Heat Exchanger**. This view changes based on the model chosen on the **Model** page.

Basic Model

When the Basic model radio button on the **Model** page is chosen the **Specs** page appears as follows.

Figure 4.24



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 the **Heat Exchanger.**

Dynamic Specification	Description
Shell/Tube Delta P	The pressure drips across the Shell/Tube side of the heat exchanger may be specified (check box active) of calculated (check box inactive).
k	Activate this option if to have the Pressure Flow k values used in the calculation of pressure drop.
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. 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.

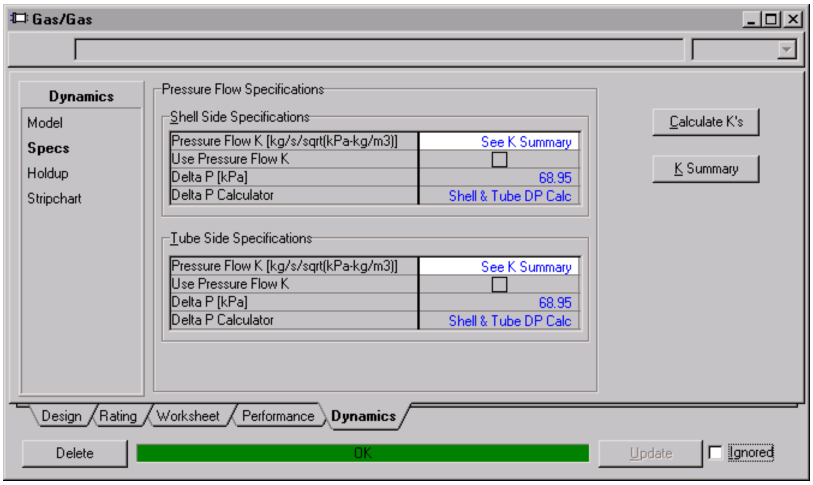
Effectively the **k Reference Flow** will result in a more linear relationship between flow and pressure drop and this is used to increase model stability during startup and shutdown where the flows are low.

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 the **Calculate k** button is clicked.

Detailed Model

When the Basic model radio button on the **Model** page is chosen the **Specs** page appears as follows.

Figure 4.25



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 pressing 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 chosen, the Shell/Tube Delta P calculator must also be set to **non specified**.
Shell/Tube Delta P	The pressure drops across the Shell/Tube side of the heat exchanger may be specified or calculated.

Dynamic Specification	Description
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 the user. • **non specified** - This option is only applicable in Dynamics 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.

Clicking the **K Summary** button will open up the **Detailed Heat Model Properties** view, refer to **Section - Detailed Heat Model Properties** for more information.

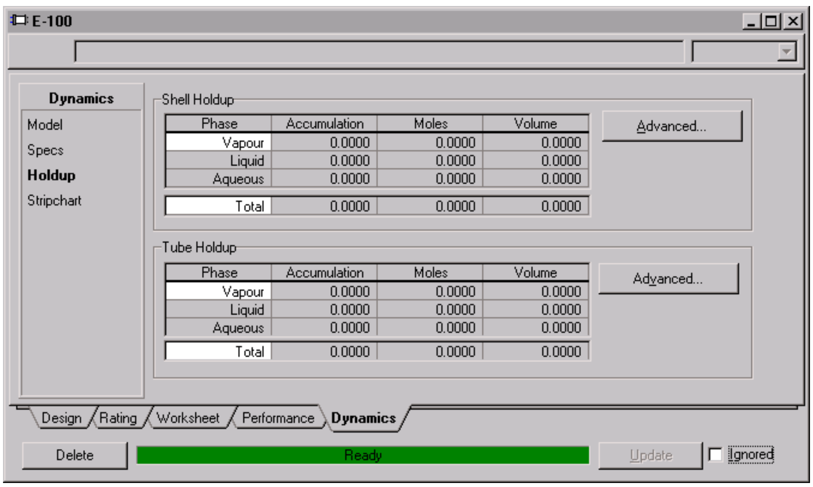
Holdup Page

The **Holdup** page contains information regarding the shell and tube holdup's properties, composition, and amount.

Basic Model

When the Basic model radio button on the **Model** page is chosen the **Specs** page appears as follows.

Figure 4.26



The **Shell Holdup** and **Tube Holdup** sections contain information regarding the shell and tube side holdup parameters. For each phase contained within the volume space of the unit operation, the following is specified:

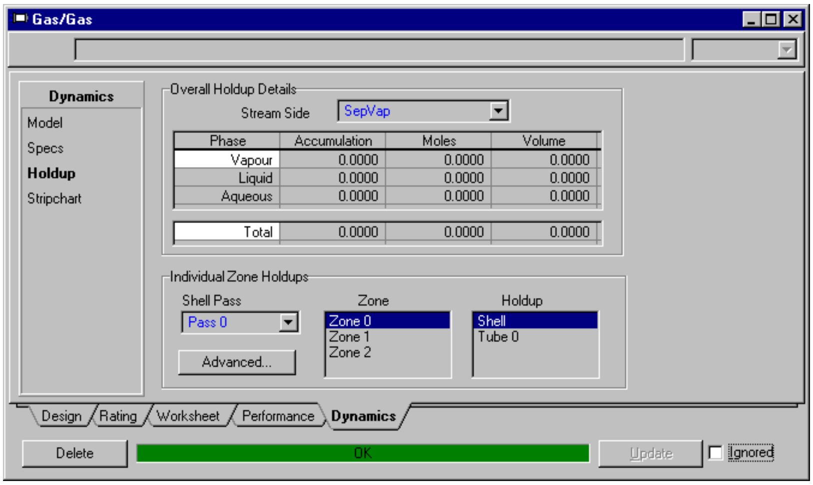
Holdup Details	Description
Accumulation	The accumulation refers to the rate of change of material in the holdup for each phase.
Moles	These cells indicate the amount of material in the holdup for each phase.
Volume	These cells indicate the holdup volume of each phase.

Click the Advanced button to accesses a view that provides more detailed information about the holdup of that unit operation. Refer to **Section 1.2.7 - Advanced Holdup Properties** for more information.

Detailed Model

When the Detailed model radio button on the **Model** page is chosen the **Specs** page appears as follows.

Figure 4.27



The **Overall Holdup Details** section contains information regarding the shell and tube side holdup parameters. For each phase contained within the volume space of the unit operation, the following is specified:

Holdup Details	Description
Accumulation	The accumulation refers to the rate of change of material in the holdup for each phase.
Moles	These cells indicate the amount of material in the holdup for each phase.
Volume	These cells indicate the holdup volume of each phase.

The **Individual Zone Holdups** section 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 and then press the **Advanced** button. To choose individual holdups you must specify the **Zone** and **Layer** in the corresponding drop down lists. Refer to **Section 1.2.7 - Advanced Holdup Properties** for more information.

Stripchart Page

The **Stripchart** page allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the **Create Stripchart** button to display the stripchart.

4.3.4 Dynamic Heat Exchanger Example

A hydrocarbon process stream will be cooled in a **Heat Exchanger** unit operation using the **Simple Rating** model. The heat exchanger will initially be built in steady state. A dynamic model of the same heat exchanger will then be created in **Dynamic** mode.

1. Create the following Fluid Package:

Property Package	Components
Peng Robinson	C1, C2, C3, i-C4, n-C4, i-C5, n-C5, C6, C7, C8

2. Create streams **Process Feed** and **Utility Feed** with the following properties:

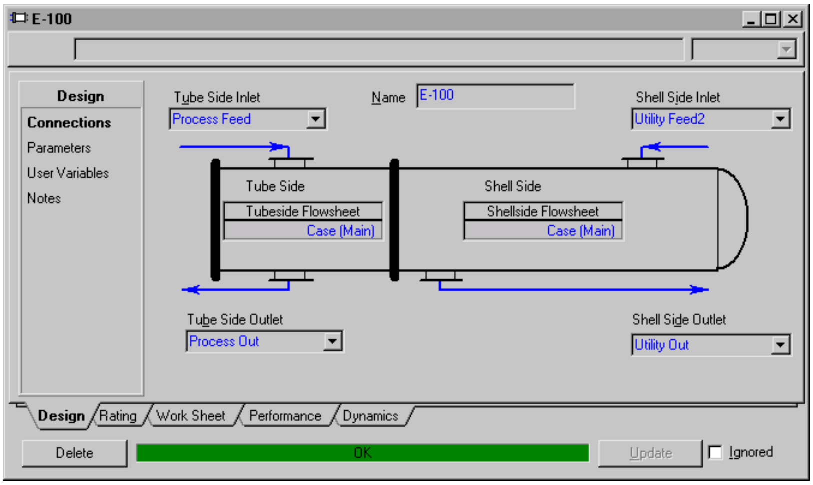
Stream Name	Process Feed	Utility Feed
Vapor Frac	1.0000	0.0000
Pressure [psia]	145.00	290.00
Molar Flow [lbmole/hr]	23.00	441.00
Comp Mole Frac [Methane]	0.1	0.00
Comp Mole Frac [Ethane]	0.1	0.01
Comp Mole Frac [Propane]	0.1	0.95
Comp Mole Frac [i-Butane]	0.1	0.04
Comp Mole Frac [n-Butane]	0.1	0.00
Comp Mole Frac [i-Pentane]	0.1	0.00
Comp Mole Frac [n-Pentane]	0.1	0.00
Comp Mole Frac [n-Hexane]	0.1	0.00
Comp Mole Frac [n-Heptane]	0.1	0.00
Comp Mole Frac [n-Octane]	0.1	0.00

3. Install a **Valve** operation with the following connections:

UNIT OPERATION TYPE	VALVE	
Tab [Page]	**Input Area**	**Entry**
Design [Connections]	Name	VLV-100
	Feed	Utility Feed
	Product	Utility Feed2
Design [Parameters]	Delta P	15 psi

4. Install the **Heat Exchanger** operation in the **Connections** page of the **Design** tab:

Figure 4.28



5. On the **Parameters** page of the **Design** tab, specify a **Steady State Rating** model in the **Heat Exchanger Model** drop down list. Enter a value of 1.45 psi for both the shell and tube side pressure drop.

6. The solution should now converge. Once the **Heat Exchanger** unit operation has finished solving, switch to **Dynamic** mode.

Dynamics Assistant Button

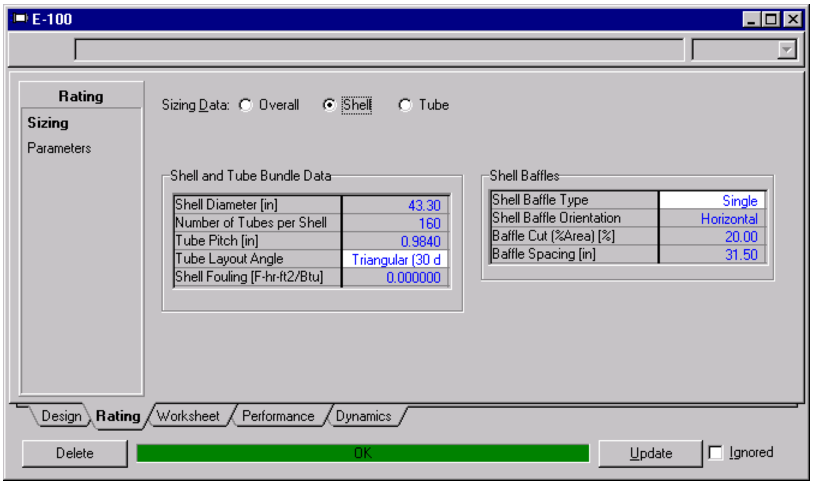
7. Press the **Dynamics Assistant** button. A list of modifications to the flowsheet will be displayed in order to run the simulation case in Dynamics. Press the ***Make Changes*** button in order to implement the suggested modifications.

8. Specify the geometry of the heat exchanger in the **Sizing** page of the **Ratings** tab. Select the **Overall** radio button. In the **Configuration** section, specify the following:

Configuration Parameter	Entry
Number of Shells in Series	1
Number of Shells in Parallel	1
Tube Passes	2
Orientation	Horizontal
TEMA Type	A - E - L

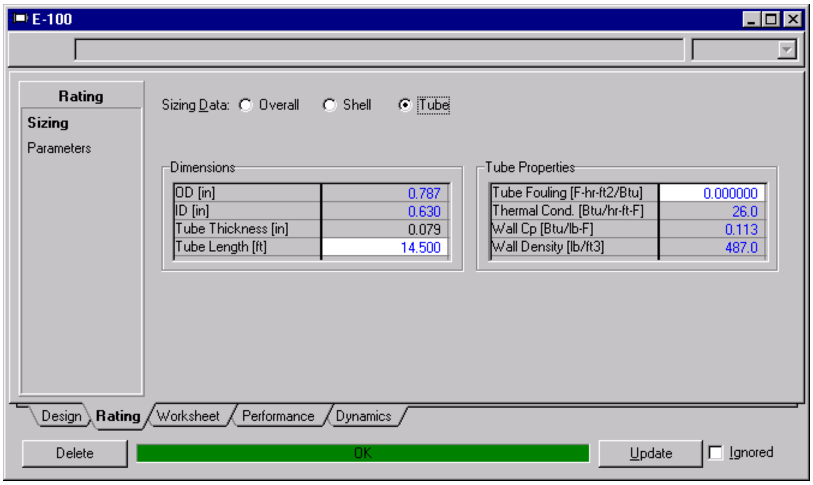
9. Select the **Shelll** radio button. Specify the rating information as shown in **Figure 4.29**:

Figure 4.29



10. Select the **Tube** radio button. Specify the rating information as shown in **Figure 4.30**. Note that if you specify both the inner and outer tube diameter, the tube thickness will be calculated:

Figure 4.30



For more information regarding the preparation of equipment for ***Dynamic*** *mode, see* ***Section 1.4.2 - Moving from Steady State to Dynamics***

Note that the shell diameter and tube length values have been modified from their default values. If effective level control is to be achieved, it is necessary to size any vessel operation, including the **Heat Exchanger**, with a reasonable liquid residence time. This exchanger has been sized with a liquid residence time of 10 minutes.

11. On the **Parameters** page of the **Rating** tab, you can choose the method used to calculate the overall Heat Transfer Coefficient, UA, and the Overall Pressure drop across the shell and tube sides of the Heat Exchanger. Select the **Detailed** model radio button.
12. In the **Shell** and **Tube HT Coefficient Calculator** cells, choose the **Shell & Tube** option.
13. In both the **Shell** and **Tube Pressure Drop Calculator** cells, choose the **not specified** option.
14. Add a **PID Controller** operation which will serve as the **Heat Exchanger** level controller. Specify the following details:

For more information on adding controllers refer to ***Chapter 10 - Logical Operations****.*

Tab [Page]	Input Area	Entry
Design [Connections]	Name	HEX LC
	Process Variable Source	E-100, d_Holduplevel_1
	Output Target Object	VLV-100
Design [Parameters]	Action	Reverse
	Kc	1
	Ti	5 minutes
	PV Minimum	0%
	PV Maximum	100%

15. From the bottom of the **PID Controller** property view, press the ***Face Plate*** button. On the **Face Plate**, change the controller mode to **Auto** on the face plate and input a setpoint of 50%.

You will require HYSYS.Plant + in order to perform your step, refer to ***Section 1.5 - Plant+****.*

In order to implement level control on the **Heat Exchanger** in HYSYS, it is necessary to set the product nozzles of all exiting streams to the top of the exchanger.

16. In the **Nozzles** page of the **Rating** tab, set the **Elevation (% of height)** to 100% for streams **Process Out** and **Utility Out**.

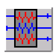
17. At this point, the **Heat Exchanger** unit operation is fully specified. Run the Integrator until all the variables line out. The **Product** stream conditions are displayed in the **Conditions** page of the **Worksheet** tab.

Figure 4.31

Name	Process Feed	Process Out	Utility Feed2	Utility Out
Vapour	1.0000	0.2732	0.0343	1.0000
Temperature [F]	298.5	143.9	130.5	133.1
Pressure [psia]	145.000	143.550	273.581	273.550
Molar Flow [lbmole/hr]	257.2	257.2	622.1	622.3
Mass Flow [lb/hr]	16750.2612	16750.2539	27693.6111	27705.3030
LiqVol Flow [barrel/day]	1930.6109	1930.6100	3734.0365	3735.6055
Molar Enthalpy [Btu/lbmole	-5.360e+04	-7.281e+04	-5.018e+04	-5.016e+04
Molar Entropy [Btu/lbmole-F	48.52	29.36	25.42	26.57
Heat Flow [Btu/hr]	-1.3783e+07	-1.6950e+07	-3.1214e+07	-2.8059e+07

4.4 LNG

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.



LNG Button

Some of the major features included 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 modelled.
- 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.

To install the **LNG** operation, press **F12** and choose **LNG** from the **UnitOps** view or select the ***LNG*** button in the Object Palette.

To ignore the **LNG**, select the **Ignore** check box. HYSYS will completely disregard the operation (and will not calculate the outlet stream) until you restore it to an active state by clearing the check box.

4.4.1 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.

$$M(H_{in} - H_{out}) + Q_{internal} + Q_{external} = \rho\frac{d(VH_{out})}{dt} \tag{4.22}$$

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:

$$f = \sqrt{density} \times k\sqrt{P_1 - P_2} \tag{4.23}$$

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) and Overall (UA) Heat Transfer Coefficients

Note that several of the pages in the LNG view indicate whether the information applies to steady state or dynamics.

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. 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, users can also specify a UA value in dynamics 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.

Note that if the user specifies the convective UA values in dynamics mode, than the size and metal holdup of the LNG are still considered.

In dynamics 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.

Ideally in dynamics mode the convective heat transfer coefficient, U, for each stream is supplied. An initial value can be estimated from correlations commonly available in the literature or from the steady state UA values. The values supplied 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 will correctly scale the U values based on the flow.

If the streams in the rating model are properly laid to optimize heat transfer (i.e. 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 will approximate the convective UA values that can be supplied in dynamics mode for the same results.

Dynamic Specifications

The following is a list of the minimum specifications required for the **LNG** unit operation to solve:

Dynamic Specifications	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 in 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 in 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 in 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 in the **Model** page of the **Dynamics** tab.

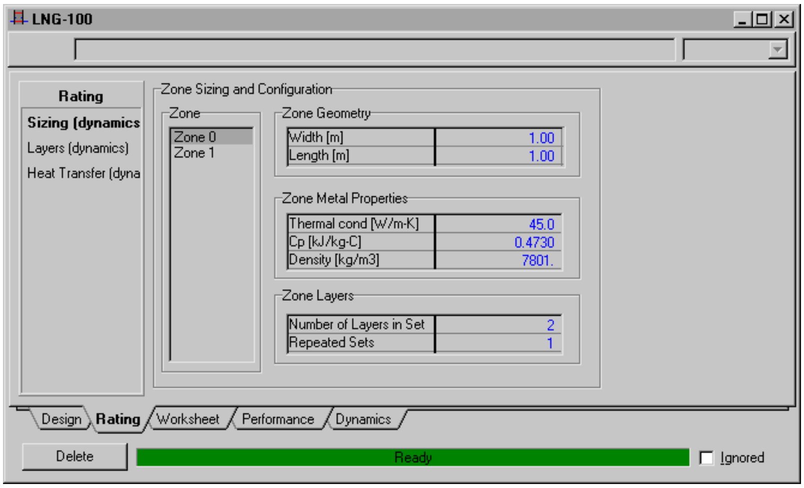
4.4.2 Rating Tab

The **Rating** tab contains three pages: the **Sizing (dynamics)**, **Layers (dynamics)**, and **Heat Transfer (dynamics)** page.

Sizing (dynamics) Page

In the **Sizing (dynamics)** page, you can specify the geometry of each zone in the **LNG** unit operation:

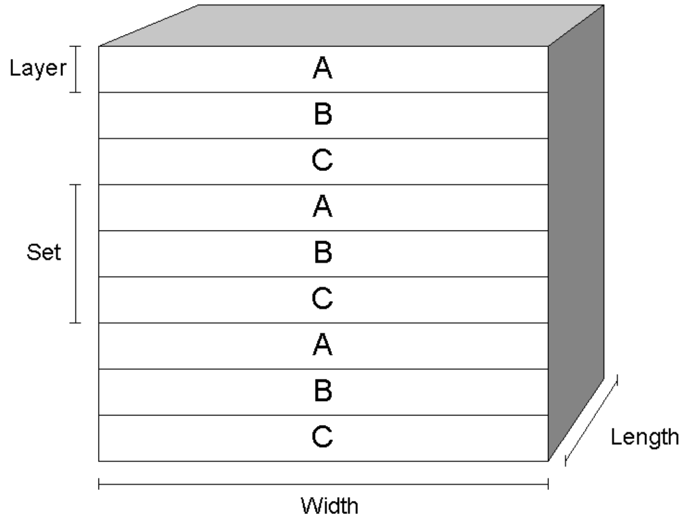
Figure 4.32



You are able to 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 may be 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. Figure 4.33 displays an **LNG** exchanger block (zone) with 3 sets, each containing 3 layers:

Figure 4.33



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, highlight the individual zone in **Zone** group and make the necessary changes to the other group boxes.

The **Zone Geometry** group box 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 box contains information regarding the metal heat transfer properties:

- Thermal Conductivity
- Specific Heat Capacity, Cp
- Density

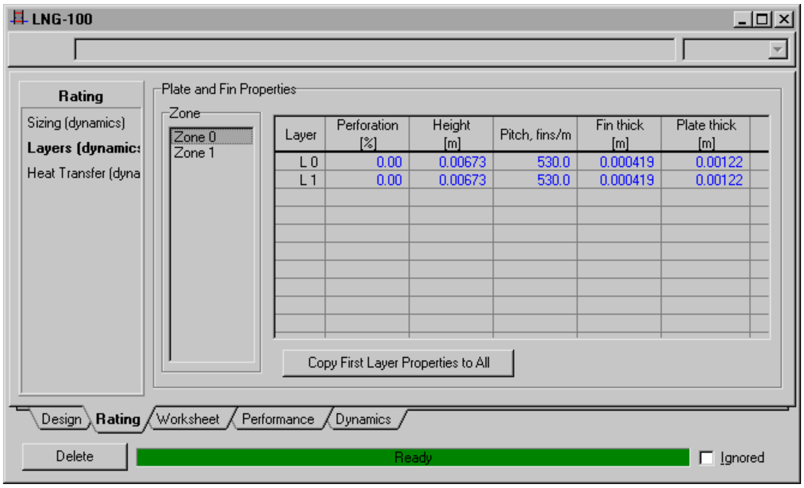
The **Zone Layers** group box contains the following information regarding the configuration of layers in the zone:

- Number of Layers in a Set
- Repeated Sets

Layers (dynamic) Page

The **Layers (dynamics)** page contains information regarding the plate and fin geometry:

Figure 4.34



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	This is 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.

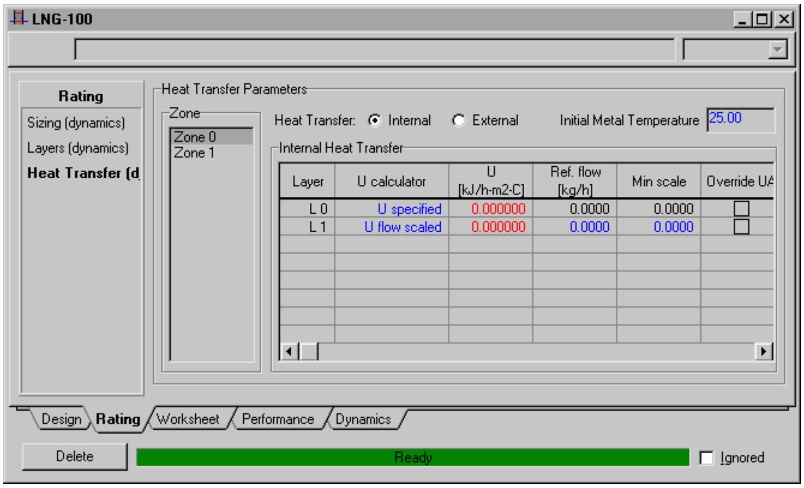
The **Copy First Layer Properties to All** button can be pressed if you wish to specify all the layers in the zone with the same plate and fin properties.

Heat Transfer (dynamics) Page

The **Heat Transfer (dynamics)** page displays the heat transfer coefficients associated with the individual layers of the **LNG** unit operation.

If the **Internal** radio button is selected, the internal heat transfer coefficient associated with each layer is displayed.

Figure 4.35



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 may be supplied for each zone in the **Initial Metal Temperature** field.

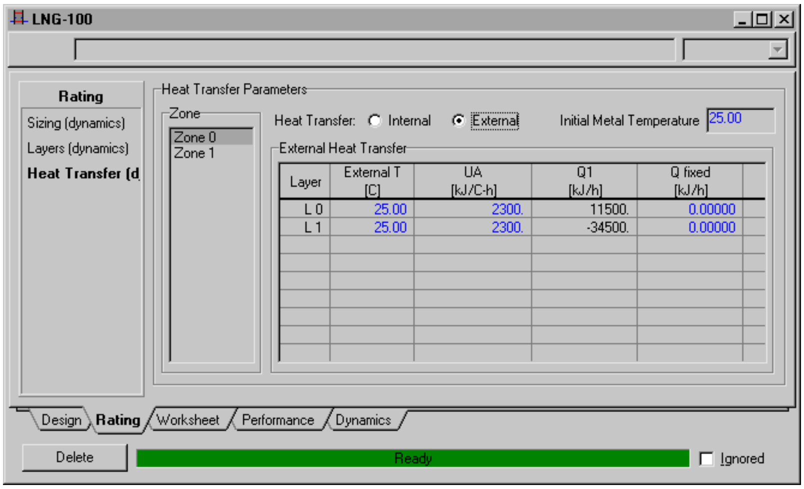
Since a repeating stacking pattern is used, the top most layer of a set is assumed to exchange heat with the bottom layer of the set above.

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 box contains the following parameters:

Internal Heat Transfer 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 will be used to calculate U.
U	The internal heat transfer coefficient is specified in this cell.
Override UA	The overall UA may be specified if the **Override UA** check box is activated. The specified UA value is used without the consideration or back calculation of the internal heat transfer coefficient, U.
UA	The overall UA is specified in this cell.

If the **External** radio button is selected, the overall UA associated with heat loss to the atmosphere is displayed.

Figure 4.36



Like the internal heat transfer coefficients, the external overall UA must be specified.The **External Heat Transfer** group box contains the following parameters:

External Heat Transfer 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 (e.g. electrical tracing). If heat is gained in the holdup, **Qfixed** is positive; if heat is lost, **Qfixed** is negative.

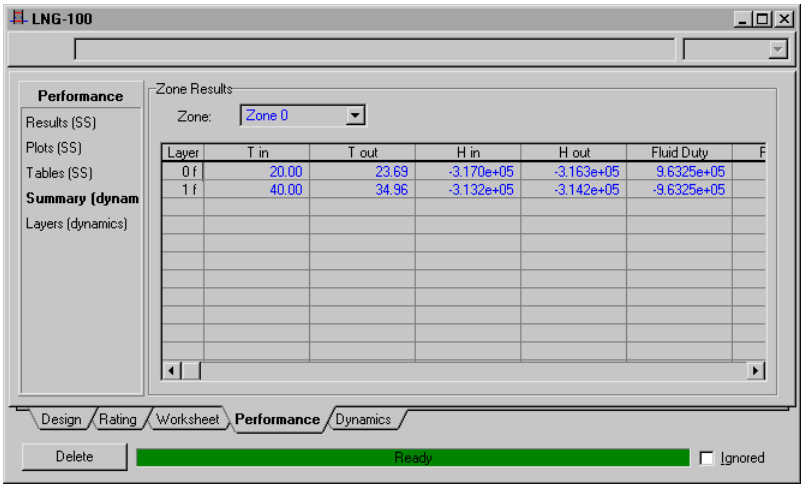
4.4.3 Performance Tab

The **Performance** tab contains two pages which pertain to the dynamic simulation of the **LNG** unit operation: the **Dynamic** and **Layers** page.

Summary Page

The **Summary** page displays the results of the dynamic **LNG** unit operation calculations.

Figure 4.37



*If the **Combined Layers** option is checked in the **Model** page of the **Dynamics** tab, some parameters in the **Dynamic** page in the **Performance** page may include contributions from multiple layers.*

In the **Summary** page, the following zone properties are displayed for each layer:

- Inlet Temperature
- Exit Temperature
- Inlet Enthalpy
- Exit Enthalpy
- Fluid Duty
- Fluid Volume
- Surface Area

The **Fluid Duty** is defined as the energy supplied 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.

Layers Page

The **Layers** page displays information regarding local heat transfer and fluid properties at endpoint locations in each layer. The information displayed in this page is not central to the performance of the **LNG** operation.

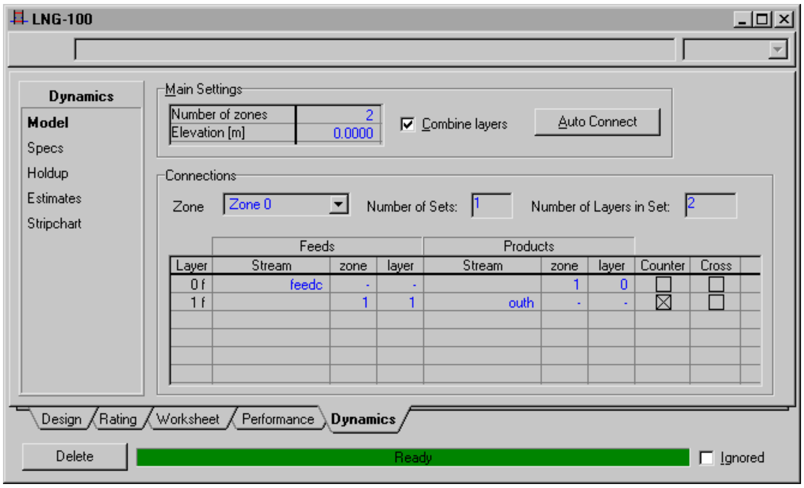
4.4.4 Dynamics Tab

The **Dynamics** tab contains five pages: the **Model**, **Specs**, **Holdup Estimates** and **Stripchart** page.

Model Page

In the **Model** page, you can specify how each layer in a multi-zone **LNG** unit operation is connected.

Figure 4.38



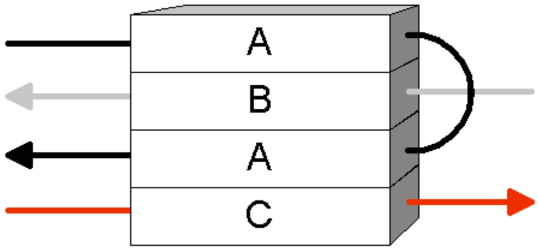
Main Settings

The **Main Settings** group box displays the following **LNG** model parameters:

Main Setting Parameter	Description
Number of Zones	The number of zones in a **LNG** unit operation can be specified in this cell.
Elevation	The elevation is significant in the calculation of static head in and around the **LNG** unit operation.
Combine Layers Check box	With the **Combine Layers** box selected, individual layers (holdups) carrying the same stream in a single zone will be calculated using a single holdup. The **Combine Layers** option increases the speed of the Dynamic solver. (The **Combine Layers** option usually yields results that are similar to a case not using the option.)

The **Connections** group box 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 may 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 4.39



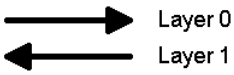
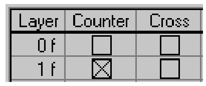
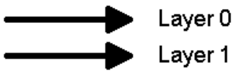
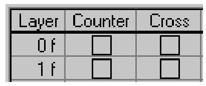
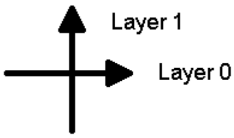
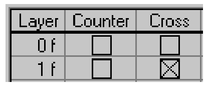
Connections Group

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 are displayed. 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

An example of a LNG operation with a counter current flow configuration is provided in ***Section 4.4.6 - Dynamic LNG Example***.

You may specify the relative direction of flow in each layer in the zone. Layers may 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** check box selected in the **Connections** group box. The following table lists three possible flow configurations:

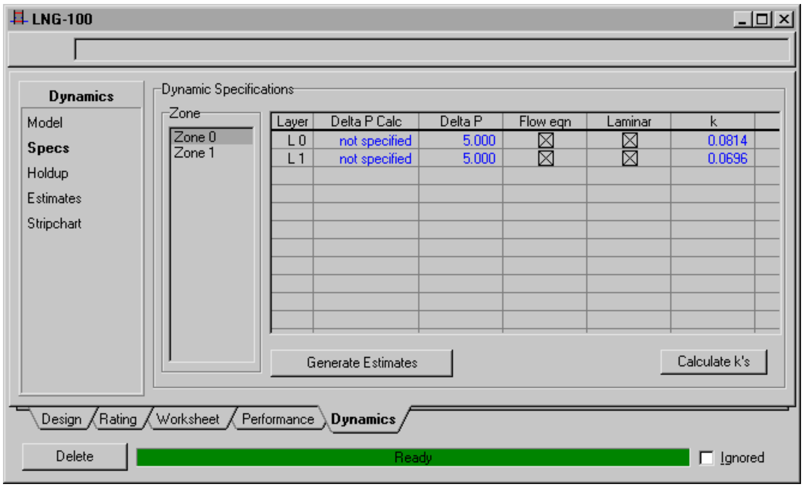
Description	Flow Direction	Flow Setting
Counter Current Flow		
Parallel Flow		
Cross Flow		

To implement counter current flow for two streams in a single exchanger block, ensure that the **Counter** check box is selected for only one of the streams. If the **Counter** check box is selected for both streams, the flow configuration will still be 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 4.40



The following parameters are displayed 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** - The pressure drop is specified by the user. • **non specified** - Pressure drop across the layer is calculated from a pressure flow relationship. You must specify a k-value and activate the **Flow Eqn** option in the **Specs** page of the **Dynamics** tab if you wish to use this **non specified** Delta P calculator.
Delta P	The pressure drop across the layer of the **LNG** operation may be specified or calculated.
Flow eqn	Activate this option if you wish to have the Pressure Flow k value used in the calculation of pressure drop. If the **Flow Eqn** option is chosen, the Delta P calculator must also be set to **non specified**.
Laminar	HYSYS is able to model laminar flow conditions in the layer. Select the **Laminar** check box if the flow through the layer is in the laminar flow regime.

*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.*

Dynamic Specification	Description
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 pressing the **Calculate k’s** button. Ensure that there is a non zero pressure drop across the **LNG** layer before the **Calculate k** button is pressed. Each zone layer can be specified with a flow and set pressure drop by pressing the **Generate Estimates** button.

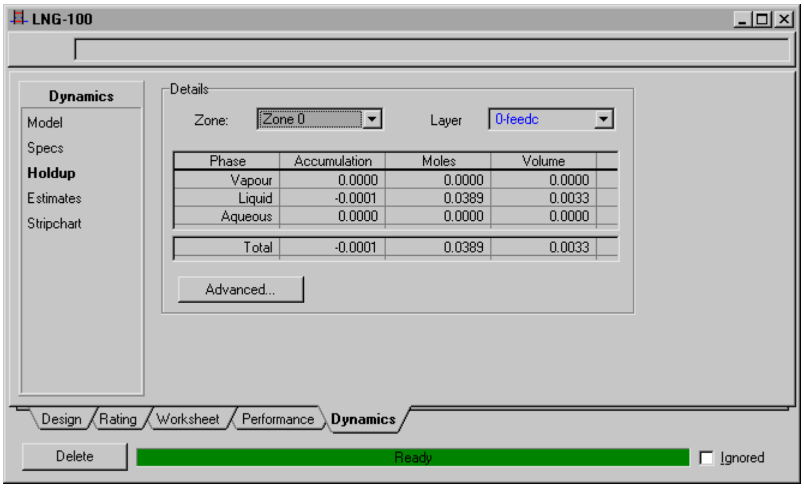
Pressing the **Generate Estimates** button calculates initial pressure flow conditions for each layer. HYSYS will generate estimates using the assumption that the flow of a particular stream entering the exchanger block (zone) will be distributed equally among the layers. The generated estimates are displayed in 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 will arise in the pressure flow solver if a stream’s flow is set in one layer and calculated in the neighbouring layer.

Holdup Page

The **Holdup** page contains information regarding each layer's holdup's properties, composition, and amount.

Figure 4.41



The **Details** section contains detailed holdup properties for every layer in each zone of the **LNG**. For each phase contained within the volume space of the unit operation, the following is specified:

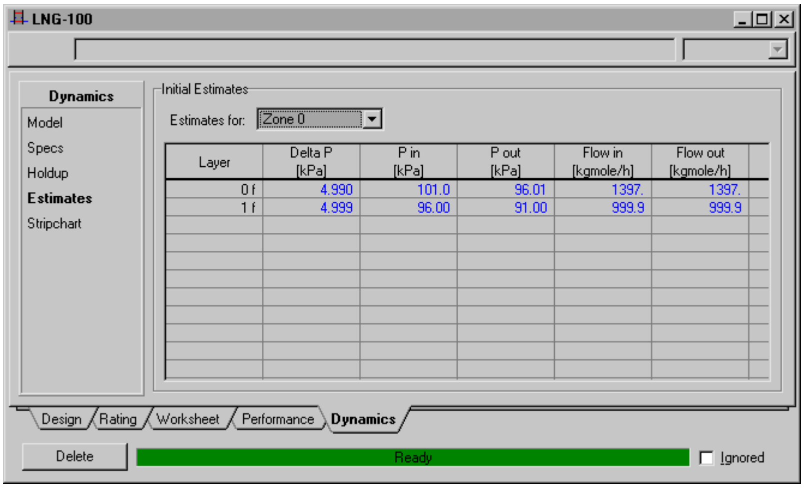
Holdup Details	Description
Accumulation	The accumulation refers to the rate of change of material in the holdup for each phase.
Moles	These cells indicate the amount of material in the holdup for each phase.
Volume	These cells indicate the holdup volume of each phase.

In order to view the advanced properties for individual holdups, you must first choose the individual holdup and then press the **Advanced** button. To choose individual holdups you must specify the **Zone** and **Layer** in the corresponding drop down lists. Refer to **Section 1.2.7 - Advanced Holdup Properties** for more information.

Estimates Page

The **Estimates** page contains pressure flow information surrounding each layer in the LNG unit operation:

Figure 4.42



The following pressure flow information is displayed in the **Estimates** page:

- Delta P
- Inlet Pressure
- Exit Pressure
- Inlet Flow
- Exit 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 pressing the **Generate Estimates** button in the **Specs** page of the **Dynamics** tab. HYSYS will generate estimates using the assumption that the flow of a particular stream entering the exchanger block (zone) will be distributed equally among the layers.

Stripchart Page

The **Stripchart** page allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the **Create Stripchart** button to display the stripchart.

4.4.5 HTFS-MUSE Tab

The **HTFS-MUSE** tab can only be used in Steady State mode. Refer to the Steady State Modelling guide for information regarding this tab.

4.4.6 Dynamic LNG Example

A hot and cold hydrocarbon stream will exchange heat in an **LNG** unit operation using the **Simple Rating** model. The **LNG** operation will be built directly in **Dynamic** mode.

1. Create the following Fluid Package:

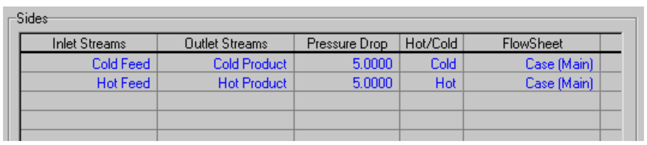
Property Package	Components
Wilson	Ethanol, 1-Pentanol

2. Enter the simulation environment and switch to **Dynamic** mode.
3. Create streams **Hot Feed** and **Cold Feed** with the following properties:

Stream Name	Hot Feed	Cold Feed
Temperature [C]	40.00	20.00
Pressure [kPa]	111.00	111.00
Molar Flow [kgmole/hr]	1000.00	1398.00
Comp Mole Frac [Ethanol]	0.5	0.8
Comp Mole Frac [1-Pentanol]	0.5	0.2

4. Install the **LNG** operation in the **Connections** page of the **Design** tab as follows:

Figure 4.43



5. On the **Model** page of the **Dynamics** tab, specify 2 zones in the **Number of Zones** field of the **Main Settings** group box.
6. Specify the dimensions of the two zones in the **Sizing** page of the **Rating** tab. Both zones are to be specified exactly the same:

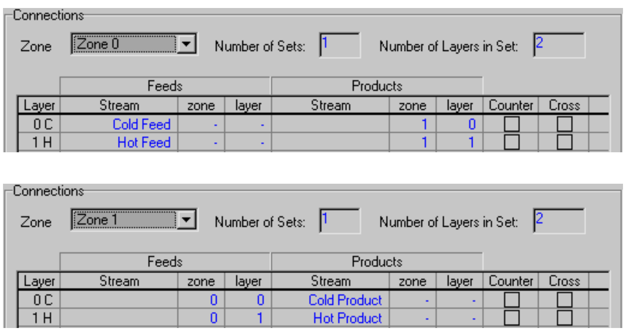
Zone Sizing and Configuration Parameter	Entry
Zone Width (m)	1.00
Zone Length (m)	1.00
Zone Thermal Conductivity (W/m-K)	160
Specific Heat Capacity (kJ/kg-C)	0.8800
Number of Layers in Set	2
Repeated Sets	1

7. Specify the plate and fin properties for each layer in both zones in the **Layers** page in the **Rating** tab. Specify both zones with the same information:

Plate and Fin Properties	Entry
Perforation (%)	0
Height (m)	0.05
Pitch (fins/m)	530
Fin Thickness (m)	0.000419
Plate Thickness (m)	0.00122

8. Select the **Internal** radio button in the **Heat Transfer** page of the **Rating** tab. Specify the U value of each layer for both zones to be 12000 kJ/h-m2-C.
9. Select the **External** radio button. Assume that there is no heat lost to the environment from the **LNG** operation. Specify the **Overall UA** for each layer in both zones to be 0 kJ/C-h.
10. Click the **Auto Connect** button on the **Model** page of the **Dynamics** tab. Ensure that the following zone connections are entered in the Connections group box:

Figure 4.44



11. On the **Specs** page of the **Dynamics** tab, specify a pressure drop of 5 kPa for each layer in both zones. Click the **Calculate k's** button. HYSYS will ask you a question, click **Yes** to calculate k values for all the layers in the **LNG** operation.
12. Specify the **Delta P Calculator** for each layer in both zones as **not specified**.
13. Open the Dynamic Assistant and click the **Make Changes** button.
14. Run the Integrator until the exit variables line out (approximately 2 integration minutes). The **LNG** unit operation is configured for Parallel flow. It is desired to change the **LNG** operation flow configuration to counter current.
15. In the **Connections** group box in the **Model** page of the **Dynamics** tab, delete the streams and zone/layer specifications for Layer 1. Complete Layer 1 as follows:

Figure 4.45

Connections

Zone: Zone 0 | Number of Sets: 1 | Number of Layers in Set: 2

	Feeds			Products				
Layer	Stream	zone	layer	Stream	zone	layer	Counter	Cross
0 C	Cold Feed	-	-		1	0	☐	☐
1 H		1	1	Hot Product	-	-	☒	☐

Connections

Zone: Zone 1 | Number of Sets: 1 | Number of Layers in Set: 2

	Feeds			Products				
Layer	Stream	zone	layer	Stream	zone	layer	Counter	Cross
0 C		0	0	Cold Product	-	-	☐	☐
1 H	Hot Feed	-	-		0	1	☒	☐

Note that counter current flow is implemented by selecting the **Counter** check box for Layer 1 in both zones and changing the feed and product streams connections. To implement counter current flow, the **Hot Feed** stream must enter Zone 1 and the **Cold Feed** stream must enter Zone 0. Observe also that the **Counter** check box is not selected for Layer 0 in either zone. Select **Counter** flow for only one stream if counter current flow is desired.

16. Run the Integrator until all the variables line out. The **Hot Product** and **Cold Product** streams are displayed in the **Conditions** page of the **Worksheet** tab.

Figure 4.46

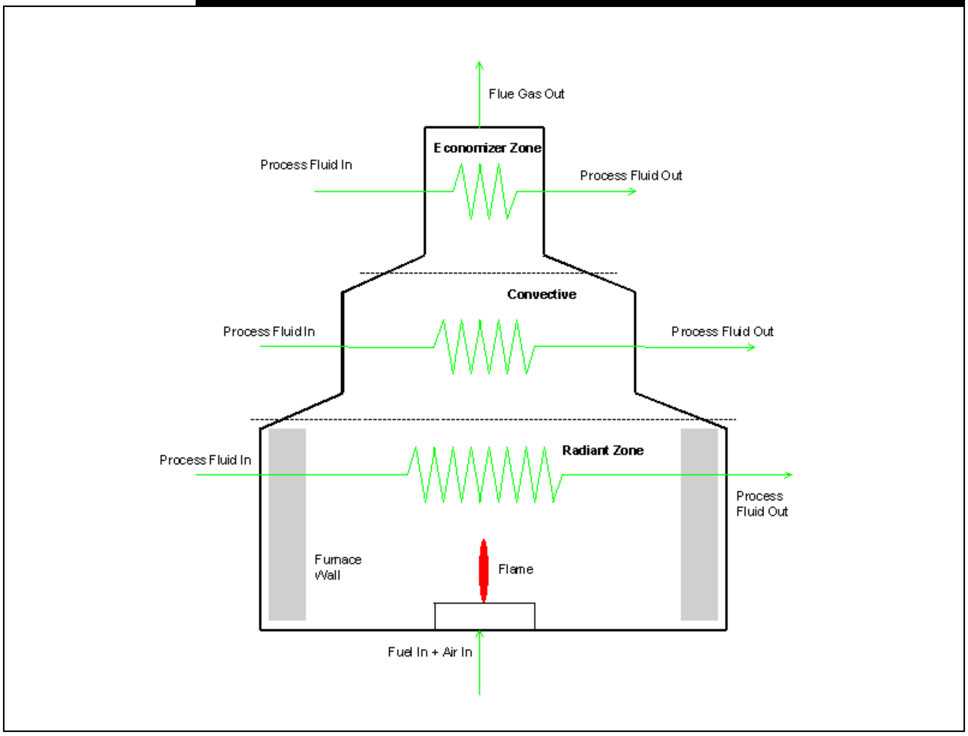
Name	Cold Feed	Hot Feed	Cold Product	Hot Product
Vapour	0.0000	0.0000	0.0000	0.0000
Temperature [C]	20.00	40.00	24.46	34.51
Pressure [kPa]	111.0	111.0	101.0	101.0
Molar Flow [kgmole/h]	1394	982.6	1394	982.6
Mass Flow [kg/h]	7.598e+004	6.594e+004	7.598e+004	6.594e+004
LiqVol Flow [m3/h]	94.56	81.28	94.56	81.28
Molar Enthalpy [kJ/kgmole]	-2.941e+005	-3.132e+005	-2.934e+005	-3.143e+005
Molar Entropy [kJ/kgmole-C	22.74	62.41	22.74	44.56
Heat Flow [kJ/h]	-4.102e+008	-3.078e+008	-4.091e+008	-3.088e+008

4.5 Fired Heater (Furnace)

This operation is available as a dynamic unit operation only.

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 following figure.

Figure 4.47



In general, a furnace can be divided into three zones:

- Radiant zone
- Convective zone
- Economizer zone

The Furnace 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.

Some of the major features included in the dynamic Furnace operation include:

- Flexible connection of process fluid associated in each furnace zone, e.g. radiant zone, convective zone or economizer zone. Different furnace 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 furnace 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, furnace 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 furnace 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.

To install the Fired Heater operation select the ***Fired Heater*** button from the Object Palette or press **F12** and choose Fired Heater from the **UnitOps** view

Fired Heater Button

To define the number of zones required by the furnace, fill in the number in **#External Passes** at **Connections** page of the **Design** tab.

To ignore the Furnace, select the **Ignore** check box. HYSYS will completely disregard the operation (and will not calculate the outlet stream) until you restore it to an active state by clearing the check box.

4.5.1 Theory

Combustion Reaction

The combustion reaction in the burner model of the furnace 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:

$$AF = \frac{\left(\frac{Mass\ of\ flow\ O_2}{\Sigma\ Mass\ flow\ of\ fuel}\right)}{Mass\ Ratio\ of\ O_2\ in\ Air} \tag{4.24}$$

The user can set the combustion boundaries, such as the maximum AF and the minimum AF, to control the burner flame. The flame will not 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 found on the **Parameters** page of the **Design** tab.

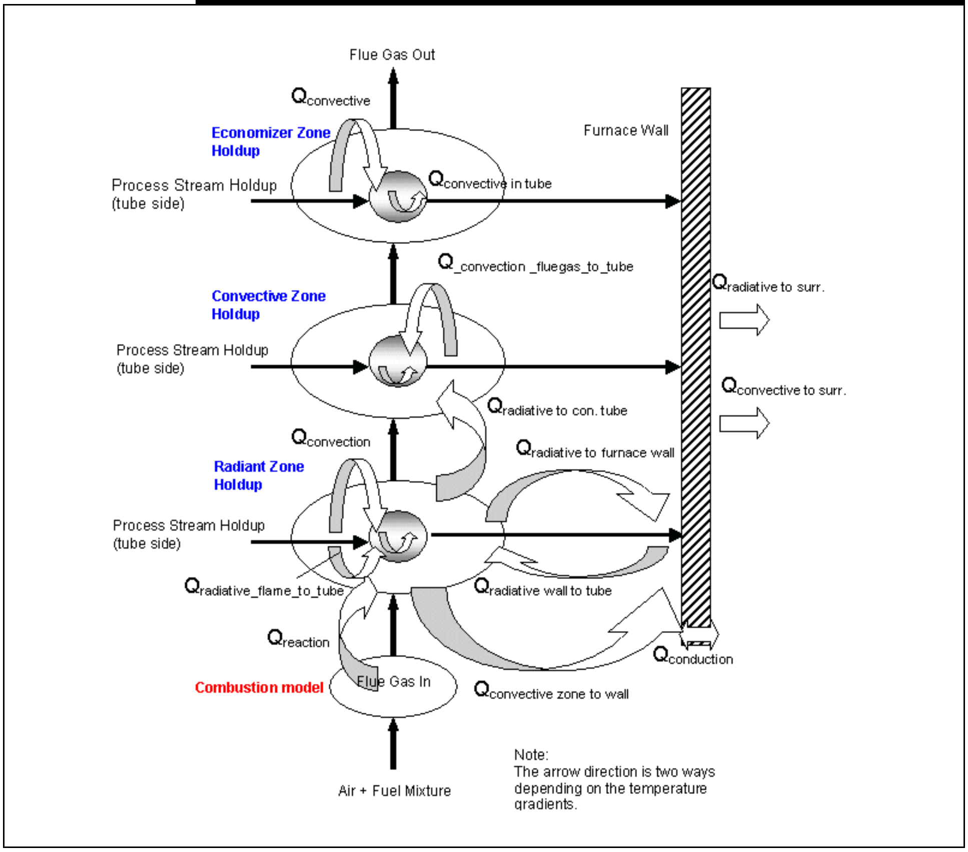
The heat released by the combustion process is the product of molar flowrate and the heat of formation of the products minus the heat of formation of the reactants at combustion temperature and pressure. In the furnace unit operation, **a traditional reaction set for the combustion reactions are not required**. You have the option to choose the fuels components (the hydrocarbons and hydrogen to be considered in the combustion reaction. You can also see the mixing efficiency of each fuel component on the **Parameter** page of the **Design** tab.

Heat Transfer

The furnace heat transfer calculations are based on energy balances for each zone. The shell side of the furnace contains five holdups: three in the radiant zone, a convective zone, and an economizer zone holdup as outlined previously in **Figure 4.47**. For the tube side, each individual stream passing through respective zones is considered as a single holdup.

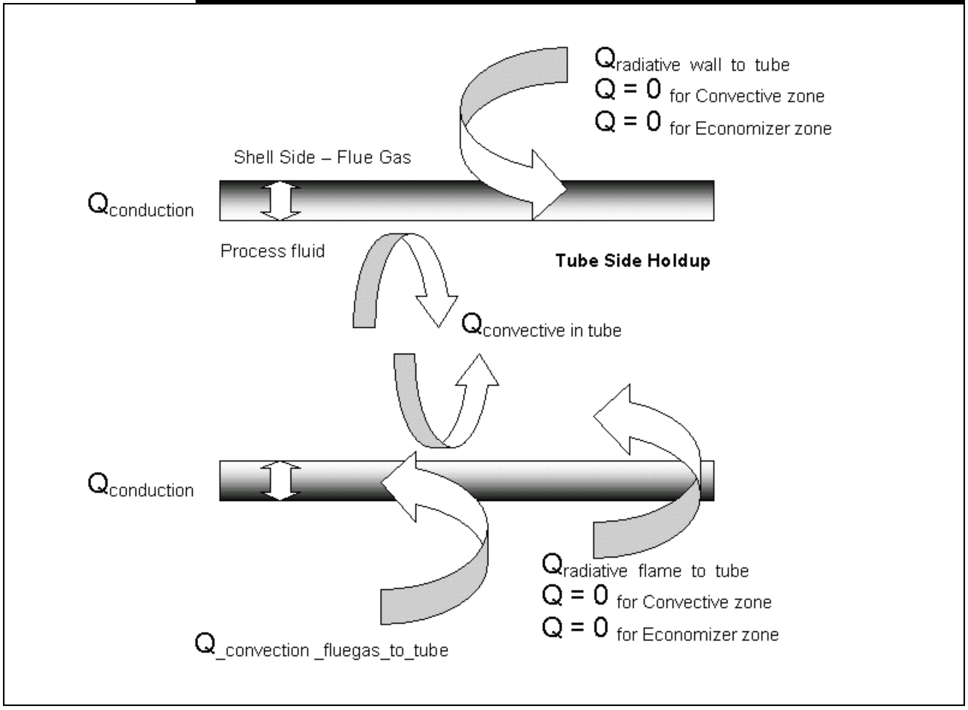
Major heat terms underlying the furnace model are illustrated in **Figure 4.49** below.

Figure 4.48



The heat terms related to the tubeside are illustrated in **Figure 4.49** below.

Figure 4.49



Taking Radiant zone as an envelope, the following energy balance equation applies:

$$\frac{d(M_{rad}H_{rad})}{dt} + \frac{d(M_{RPFTube}H_{RPFTube})}{dt}$$
$$= (M_{RPF}H_{RPF})_{IN} - (M_{RPF}H_{RPF})_{OUT} + (M_{FG}H_{FG})_{IN} \tag{4.25}$$
$$-(M_{FG}H_{FG})_{OUT} - Q_{RadToCTube} - Q_{rad\ wall\ sur} - Q_{con\ wall\ sur}$$
$$+ Q_{rad\ wall\ to\ tube} - Q_{con\ to\ wall} + Q_{reaction}$$

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 furnace wall in radiant zone to surrounding*

$Q_{con\_wall\_sur}$ = *Convective heat loss of furnace wall in radiant zone to surrounding*

$Q_{rad\_wall\_to\_tube}$ = *Radiant heat from inner furnace wall to radiant zone's tube bank*

$Q_{rad\_flame\_wall}$ = *Radiant heat from flue gas flame to inner furnace wall*

$Q_{con\_to\_wall}$ = *Convective heat from flue gas to furnace 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:

$$Q_{radiative} = \delta A \varepsilon (T_1^{\ 4} - T_2^{\ 4}) \tag{4.26}$$

where:

δ = *Stefan-Boltzmann constant, 5.669x10^{-8} W/m^{2}K^4*

ε = *emmisivity, (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:

$$Q_{convective} = UA(T_1 - T_2) \tag{4.27}$$

where:

U = *overall heat transfer coefficient, W/m^{2}K*

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:

$$U_{used} = U_{specified}\left(\frac{Mass\ flow\ at\ time\ t}{Reference\ Mass\ flow}\right)^{0.8} \tag{4.28}$$

Where $U_{specified}$ is the 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 Furnace operation, the minimum flow scaled factor can be expressed only as a positive value.

For instance if minimum flow scaled factor is +0.001 (0.1%), when this mass flow ratio is achieved, the U_{used} will stay as a constant value. Therefore,

$$U_{used} = U_{specified}(0.001)^{0.8} \tag{4.29}$$

Conductive Heat Transfer

Conductive heat transfer in a solid surface is given as:

$$Q_{conductive} = -kA\frac{(T_1 - T_2)}{\Delta t} \tag{4.30}$$

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 Furnace 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 furnace pass, a *k* value is used to relate the frictional pressure drop and molar flow, F through the furnace. This relation is similar to the general valve equation:

$$F = k\sqrt{\rho(P_1 - P_2)} \tag{4.31}$$

This general flow equation uses the pressure drop across the furnace 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:

1. 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.
2. If the flow of the system is smaller than the k_{ref}, the *k* value is given by:

$$k_{used} = k_{user\ specified} \times Factor \tag{4.32}$$

where:

Factor 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:

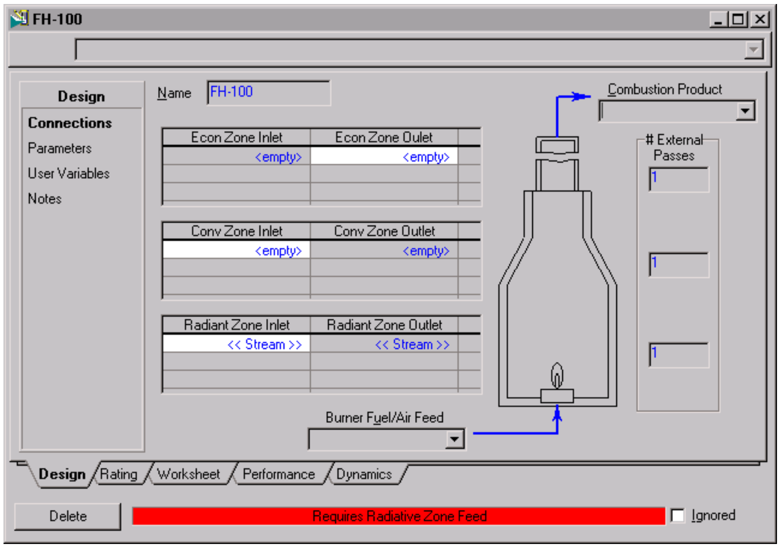
Dynamic Specifications	Description
Connections	At least one *radiant zone* inlet stream and the respective outlet zone, one burner fuel/air feed stream and one combustion product stream must be defined. There is a minimum of one inlet stream and one outlet stream required per zone. Complete the connections group for each zone of the **Design** tab.
(Zone) Sizing	The dimensions of the tube and shell in each zone in the furnace must be specified. All information in the **Sizing** page of the **Rating** tab must be completed.
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 in Integrator view is checked.
Pressure Drop	Either specify an overall delta P or an overall K value for the furnace. Specify the pressure drop calculation method on the **Tube Side PF** and **Flue Gas PF** pages of the **Dynamics** tab.

4.5.2 Design Tab

Connections Page

The **Connections** page is used to name the operation and specify inlet and outlet streams to the operation.

Figure 4.50

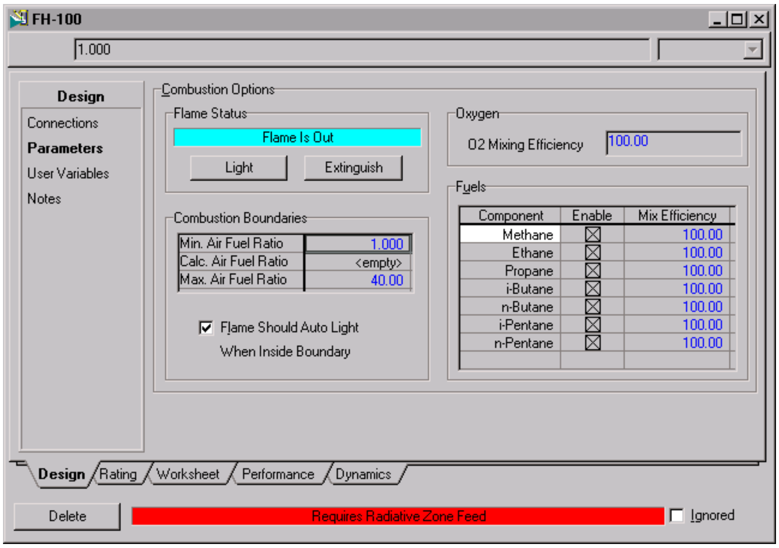


Object	Description
Econ Zone Inlet/Outlet	Allows you to specify multiple inlet and outlet streams for the Economizer zone.
Conv Zone Inlet/Outlet	Allows you to specify multiple inlet and outlet streams for the Convective zone.
Radiant Zone Inlet/Outlet	Allows you to specify multiple inlet and outlet 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	Defines the number of zones required by the furnace

Parameters Page

The **Parameters** page is used to specify the furnace combustion options.

Figure 4.51



This page is divided into four groups. The **Flame Status** group, along with displaying the flame status, allows you to toggles 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 check-box, 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

The **User Variables** page allows you to attach code and customize your HYSYS simulation case by adding User Variables. For more information on implementing this option, see the **User Variables** chapter in the **Customization Guide**.

Notes Pages

The **Notes** page provides a text editor where you can record any comments or information regarding the **HEAT EXCHANGER** or pertaining to your simulation, in general.

4.5.3 Rating Tab

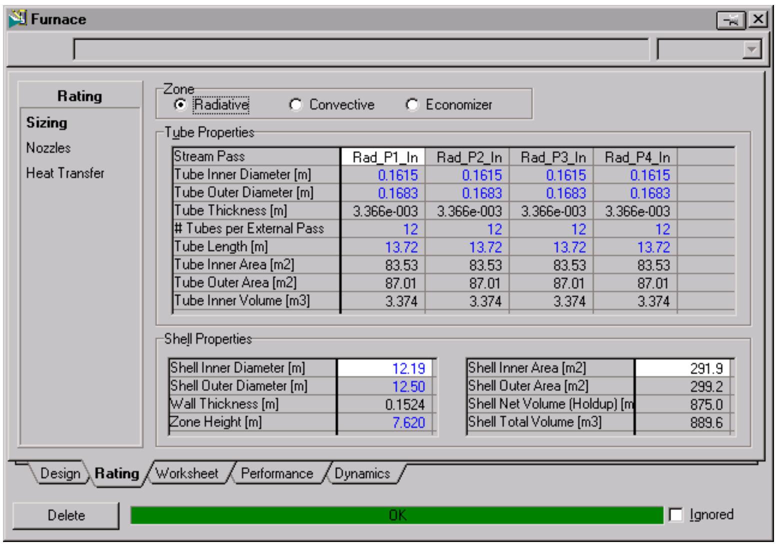
The rating tab contains three pages: **Sizing**, **Nozzles**, and **Heat Transfer**. Each page is discussed in the following sections.

Sizing Page

On the **Sizing** page, you can specify the geometry of the *radiant*, *convective* and *economizer* zones in the furnace.

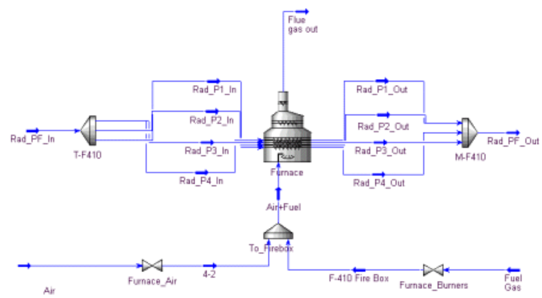
From the **Zone** group on the **Sizing** page, you can select between **Radiative**, **Convective** and **Economizer** zone views depending upon which radio button is selected. These views contain information regarding the tube and shell properties. To edit or input parameters within these views, click the individual cell and make the necessary changes.

Figure 4.52



The following figure illustrates an example of furnace setup with one radiant zone/firebox only with four tube passes. This is the simplest type.

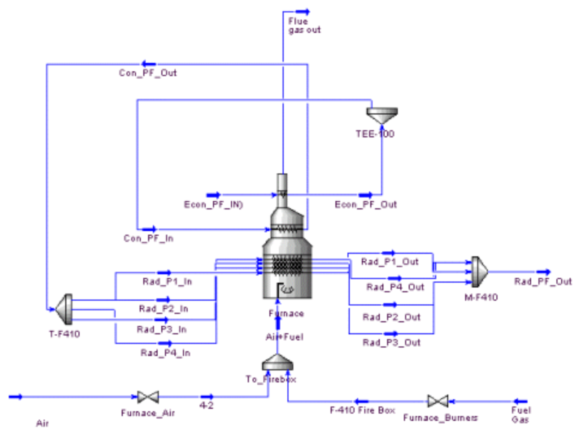
Figure 4.53



One Radiant Zone/Firebox, 4 Tube Passes

The next figure (**Figure 4.54**) shows an example of furnace setup with a radiant, convective and economizer section.

Figure 4.54



One Firebox with convective and economizer section

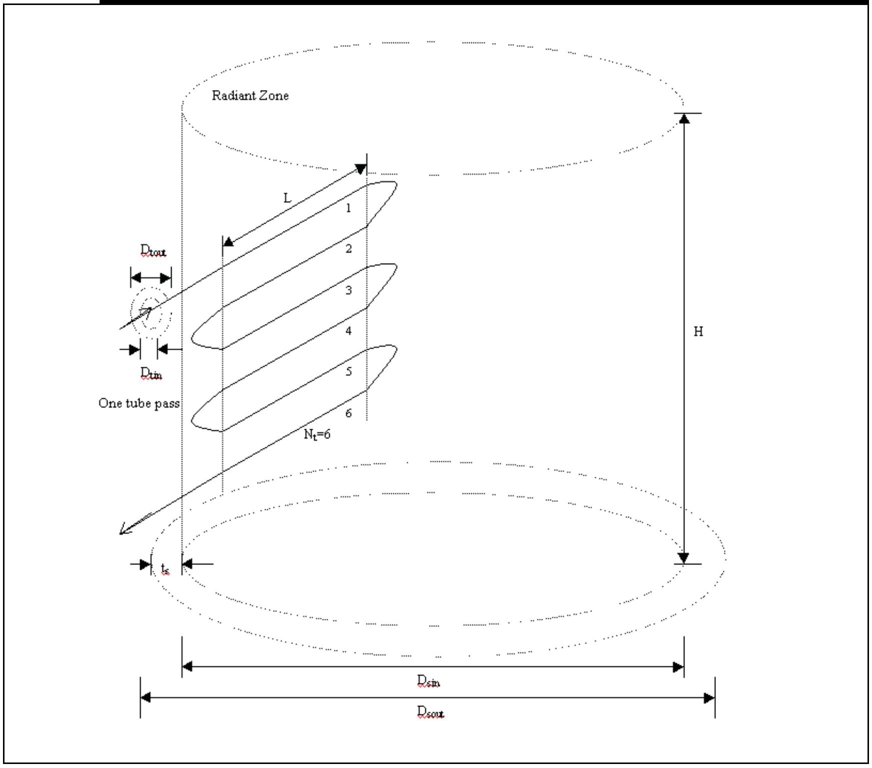
Tube Properties Group

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
- # of tubes, N_t

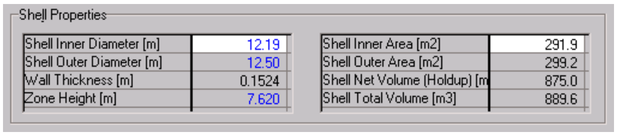
A pass in the furnace is defined as a path where the process fluid flow through a distinctive inlet nozzle and outlet nozzle. **Figure 4.55** illustrates the various dimensions of the tube and shell.

Figure 4.55



Shell Properties Group

Figure 4.56



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

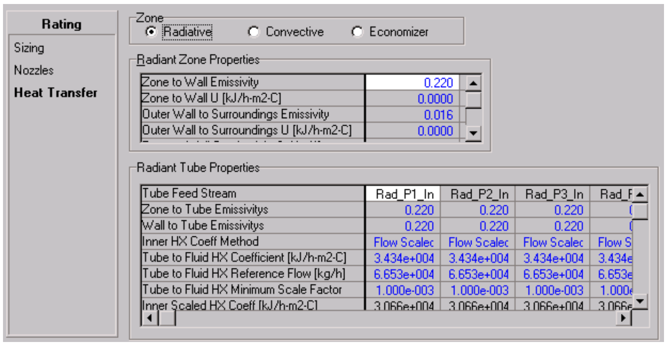
Figure 4.57

The **Nozzle** page allows you to define the base elevation to ground level of the furnace.

Heat Transfer Page

This page displays the radiant heat transfer properties, heat transfer coefficients of the furnace wall and tube and the shell and tube area and volume in each individual zone.

Figure 4.58



HYSYS accounts for the convective, conduction and radiative heat transfer in the radiant zone. For the convective heat transfer calculation, the user has two options:

- **User Specified** - the user specifies 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 (4.28)**

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 will calculate the heat transfer coefficient from the geometry/configuration of the furnace. 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	This refers to the emissivity of flue gas. HYSYS uses a constant value.
Zone to Wall U	This refers to the convective heat transfer coefficient of the radiative zone to the furnace inner wall.
Outer Wall to Surrounding Emissivity	This refers to the emissivity of the furnace outer wall.
Outer Wall to Surroundings U	This refers to the convective heat transfer coefficient of the furnace outer wall to ambient.
Furnace Wall Conductivity/ Specific Heat/ Wall Density	These are user specified properties of a single layer of furnace 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 furnace wall emissivity to the radiant zone tubes.
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 Minimum Scale Factor	This represents 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 will be smaller than U specified.
Tube to Fluid HX Coefficient	This refers to the heat transfer coefficient of the tube to the process fluid.
Tube to Fluid HX Reference Flow (kg/hr)	This refers to the mass flow at which the tube to fluid HX coefficient is based on. Usually the ideal steady state flow is recommended as input.

Tube Properties	Description
Inner Scaled HX Coefficient	Represents 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	The 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	This represents the 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 will be set to zero.
Outer Scaled U	Represents the actual HX 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 furnace 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 furnace convective heat loss calculation, Equation (4.27) if the total heat loss via furnace wall is known. The total heat loss is normally expressed as a percentage of total furnace duty. A 3-5% heat loss is an acceptable estimate.

Estimating Zone to Wall U requires trial and error techniques. One could try to input a value of U then observe the temperature profile of the flue gas exiting the radiant zone.

4.5.4 Worksheet Tab

*To see the stream parameters broken down per stream phase, open the **Worksheet** tab of the **Stream Property View**.*

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. The **Conditions, Properties,** and **Composition** pages contain selected information from the corresponding pages of the **Worksheet** tab for the stream property view. The **PF Specs** page contains a summary of the stream property view **Dynamics** tab.

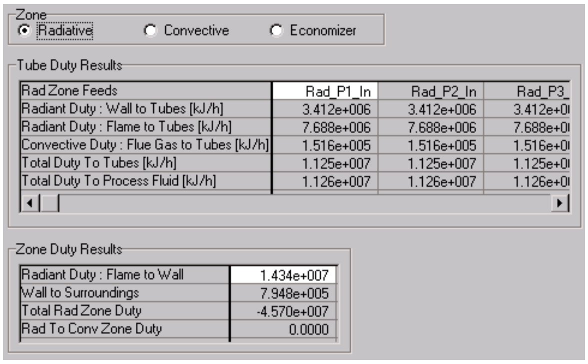
4.5.5 Performance Tab

The performance tab contains three pages which highlight the calculated temperature, duty, and pressure of the furnace operation.

Duty Page

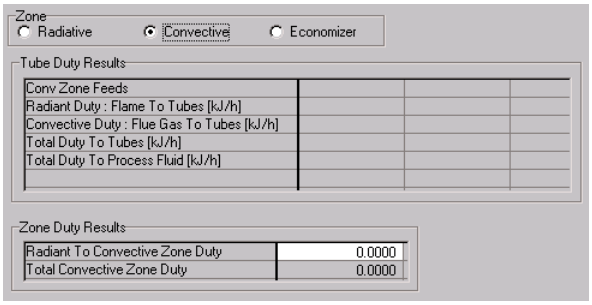
The duty page displays the results of the furnace energy balance calculation. If the **Radiative** radio button is selected, the following parameters from the Tube Duty Results group and the Zone Duty Results group are displayed:

Figure 4.59



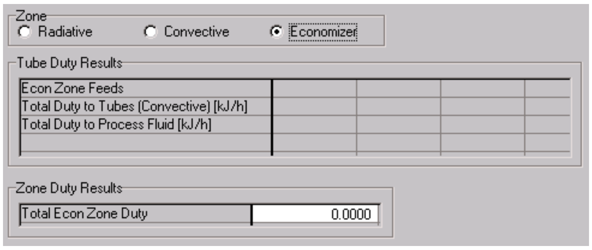
If the **Convective** radio button is displayed, the following parameters from the Tube Duty Results group and the Zone Duty Results group are displayed:

Figure 4.60

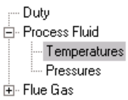


If the **Economizer** radio button is selected, the following parameters from the Tube Duty results group and Zone Duty results group are displayed:

Figure 4.61



Sub pages on the Process Fluid page.



Process Fluid Page

The **Process Fluid** page contains two sub pages: **Temperatures** and **Pressures**.

In the **Temperatures** sub page, the following parameters are displayed:

- Inlet Temp, Inlet stream process fluid temperature
- Outlet Temp, Outlet stream process fluid temperature
- Tube Inner Temp, Tube inner wall temperature

In the **Pressures** subpage, the following parameters are displayed:

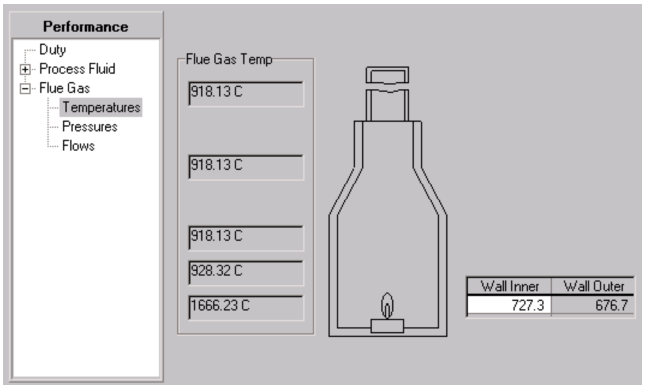
- Inlet pressure, inlet stream pressure
- Friction Delta P, friction pressure drop across the tube
- Static Head Delta P, static pressure of the stream
- Outlet Pressure, outlet stream pressure

Flue Gas Page

The **Flue Gas** page consists of three sub pages: **Temperatures**, **Pressures**, and **Flows**.

On the **Temperatures** subpage you can view your flue gas temperature and furnace inner/outer wall temperatures.

Figure 4.62

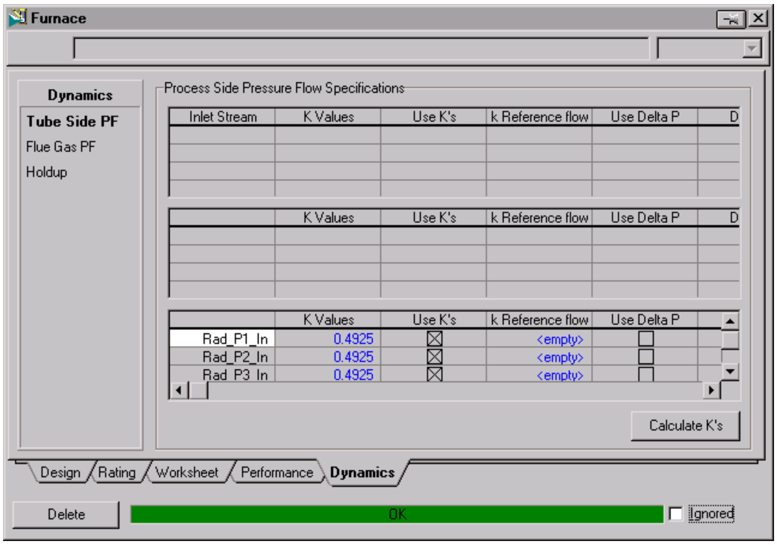


Similarly the **Pressures** sub page displays flue gas pressures, frictional delta P and static head delta P. The **Flow** sub page displays flue gas molar/mass flow.

4.5.6 Dynamics Tab

The **Dynamics** tab contains information pertaining to pressure specifications for he dynamic calculations. It contains three pages: **Tube Side PF**, **Flue Gas PF**, and **Holdup**. The following figure displays the **Tube Side PF** page.

Figure 4.63



Tube Side PF Page

On this page, you can specify how the pressure drop in each pass is calculated. The following outlines the tube side PF options available on this page.

Option	Description
Use K's?	If this box is checked, the K method will be used to calculate Delta P across the pass.
Use Delta P Spec?	If this box is checked, the pressure drop is fixed at this specified value.
Calculate K's	If this button is clicked, HYSYS will calculate the K required to maintain a specified Delta P across a defined flow condition.

Flue Gas PF Page

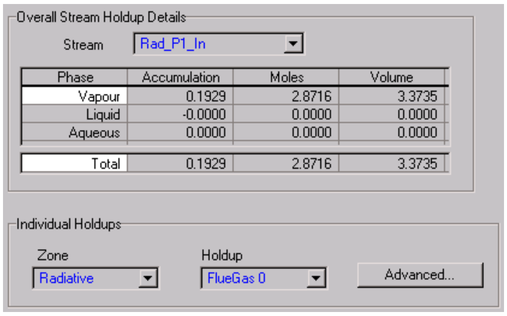
On this page, you can specify how the pressure drop in each pass is calculated. The following outlines the tube side PF options available on this page.

Option	Description
Use PF K's	If this box is checked, the K method will be used to calculate Delta P across the pass.
Use Delta P	If this box is checked, the pressure drop is fixed at this specified value.
Calculate K's	If this button is clicked, HYSYS will calculate the K required to maintain a specified Delta P across a defined flow condition.

Holdup Page

This page contains information regarding each stream's holdup properties and composition.

Figure 4.64



The Overall Stream Holdup Details group contains information regarding the stream's holdup with its associated phases. For each phase contained within the volume space of the unit operation, the following is specified:

Holdup Details	Description
Accumulation	The accumulation refers to the rate of change of material in the holdup for each phase.
Moles	These cells indicate the amount of material in the holdup for each phase.
Volume	These cells indicate the holdup volume of each phase.

Click the Advanced button to accesses a view that provides more detailed information about the holdup of that unit operation. Refer to **Section 1.2.7 - Advanced Holdup Properties** for more information.

4.6 References

[1] Perry, R.H. and D.W. Green. Perry's Chemical Engineers' Handbook (Seventh Edition) McGraw-Hill (1997) p. 11-33

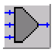
[2] Perry, R.H. and D.W. Green. Perry's Chemical Engineers' Handbook (Seventh Edition) McGraw-Hill (1997) p. 11-42

[3] Kern, Donald Q. Process Heat Transfer McGraw-Hill International Editions: Chemical Engineering Series, Singapore (1965) p. 139

5 Piping Equipment

5.1 Mixer

Under normal operation, the dynamic **MIXER** operation combines two or more inlet streams to produce a single outlet stream. 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.



Mixer Button

To install the **MIXER** operation, press **F12** and choose **MIXER** or select the ***Mixer*** button in the Object Palette.

5.1.1 Rating Tab

Nozzles Page

The **Nozzles** page contains information regarding the elevation and diameter of the nozzles. The Plant+ license is required to use the Nozzle features found on this page. Refer to **Section 1.5 - Plant+** for more information.

It is strongly recommended that the elevation of the inlet and exit nozzles are equal for this unit operation. If you wish to model static head, the entire piece of equipment can be moved by modifying the **Base Elevation relative to Ground** cell.

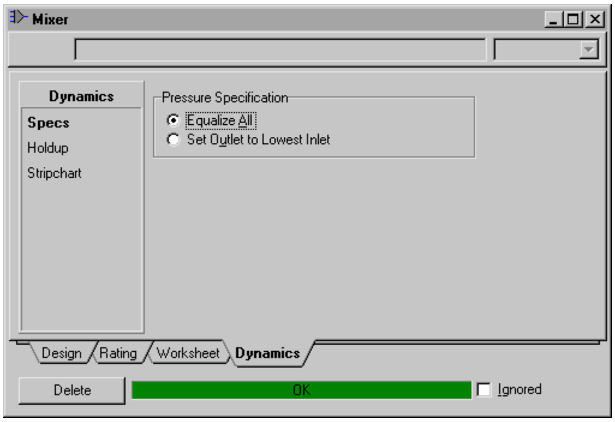
5.1.2 Dynamics Tab

The **Dynamics** tab provides access to two pages: the **Specs** and **Holdup** pages.

Specs Page

The dynamic specifications of the mixer can be specified on the **Specs** page:

Figure 5.1



In **Dynamic** mode, there are two possible dynamic specifications you can choose to characterize the mixer operation.

If the **Equalize All** radio button is selected, 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 the **Set Outlet to Lowest Inlet** radio button is selected, 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** box is unchecked. If flow reverses in the mixer, it essentially acts like a dynamic tee with the **Use Splits as Dynamic Specs** box unchecked. In dynamics, these two unit operations are very similar.

Holdup Page

*The volume of the **MIXER** is assumed to be zero.*

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.

Click the Advanced button to accesses a view that provides more detailed information about the holdup of that unit operation. Refer to **Section 1.2.7 - Advanced Holdup Properties** for more information.

Stripchart Page

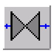
The **Stripchart** page allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the Create Stripchart button to display the stripchart.

5.2 Valve

There are 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 included 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 modelled.
- 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.



Valve Button

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.

To install the **VALVE** operation, press **F12** and choose **Valve** from the **Unit Ops** view or select the **Valve** button in the Object Palette.

To ignore the **VALVE** operation during calculations, activate the **Ignored** check box. HYSYS will disregard the operation until you clear the check box.

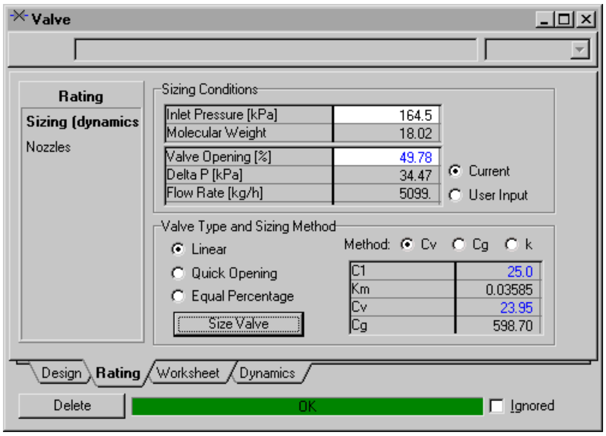
5.2.1 Rating Tab

The **Rating** tab provides access to two pages: the **Sizing** page and the **Nozzles** page.

Sizing (dynamics) Page

On the **Sizing (dynamics)** page, the sizing conditions, the valve type, and the sizing method are displayed

Figure 5.2



Sizing Conditions

The current stream conditions of the valve may be displayed by clicking on the **Current** radio button. HYSYS uses the stream conditions provided in this field to calculate valve parameters, which are used in the valve equation. If you wish to input your own sizing conditions, the **User Input** radio button should be selected.

The following stream conditions are displayed and cannot be modified:

- Inlet Pressure
- Molecular Weight

The following valve and stream conditions are displayed and can be modified in the sizing calculation:

- Valve % Opening
- Delta P
- Mass Flow Rate

Valve Type

In HYSYS, three different types of valves may be specified using the Valve operation:

- Linear
- Quick Opening
- Equal Percentage

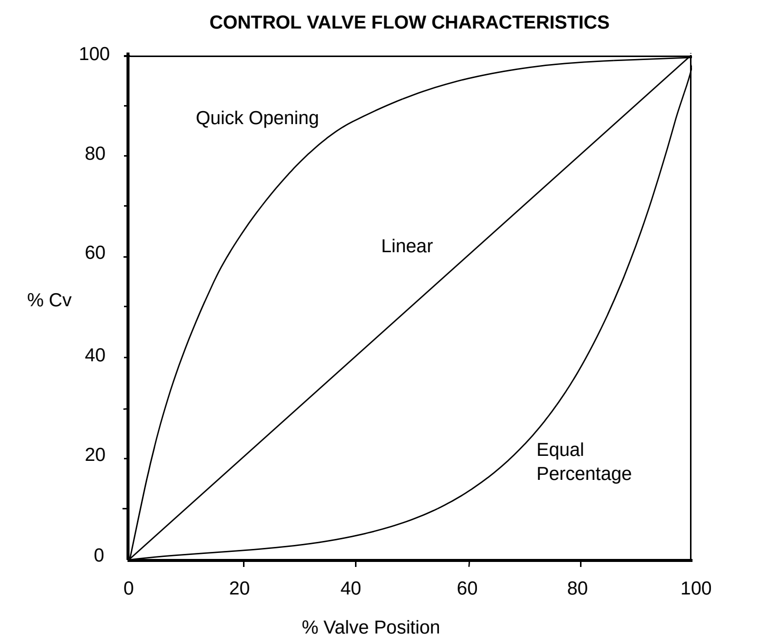
The flow rate through a control valve depends on the actual valve position. If the flow can be expressed in terms of $\%C_v$, (0% representing no flow conditions and 100% represented maximum flow conditions) then the valve characteristics of a control valve is defined as the dependence on the quantity of %Cv as a function of the actual valve percent opening.

In HYSYS, you can specify three different valve characteristics for any control valve in the simulation case.

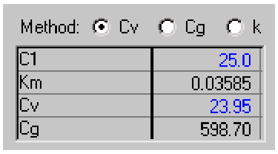
Valve Type	Description
Linear	A control valve with linear valve characteristics has a flow which is directly proportional to the valve % opening.
Quick Opening	A control valve with quick opening valve characteristics obtains larger flows initially at lower valve openings. As the valve opens further, the flow increases at a smaller rate.
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 valve characteristics are shown graphically in **Figure 5.3**:

Figure 5.3



Sizing Method



Cv Method Parameters

Three different sizing methods can be implemented in HYSYS. You can use either a Cv, Cg or k-type resistance equation to calculate flow.

If the **Cv** or **Cg** method is chosen, two of the following parameters may be specified in order to fully define the valve

- C1, Km, Cv, and Cg

C1 and *Km* cannot both be specified by the user.

If desired, a single valve parameter can be specified while HYSYS calculates the remaining parameters from the stream and valve conditions displayed in the **Sizing Conditions** group. This is achieved by pressing the ***Size Valve*** button. HYSYS provides a *C1* default value of 25.

The *Cv* and *Cg* methods calculate the flow through the valve using the following equation:

$$f(lb/hr) = 1.06C_g\sqrt{\rho(lb/ft^3)\times P_1}\times\sin\left(\frac{59.64}{C_1}\sqrt{1-\frac{P_2}{P_1}}\times cpfac\right) \tag{5.1}$$

where:

$$cpfac = \sqrt{\frac{0.4839}{1-\left(\frac{2}{1+\gamma}\right)^{\left(\frac{\gamma}{\gamma-1}\right)}}} \tag{5.2}$$

$$\gamma = C_p/C_v \tag{5.3}$$

$$Km = 0.001434C1 \tag{5.4}$$

$$C1 = \frac{Cg}{Cv} \tag{5.5}$$

where: P_1 = the pressure of the inlet stream

P_2 =the pressure of the exit stream without static head contributions

HYSYS reports the full Cv (at 100% open, which remains fixed) plus the valve opening. If the valve is 100% open then you will 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 Cv (varies with the valve opening) versus the value opening.

If the *k* method is chosen, you can either specify k or have it calculated from the stream and valve conditions displayed in the **Sizing Conditions** field. HYSYS can calculate k by pressing the ***Size Valve*** button. The *k* method calculates the flow through the valve using the following equation:

$$f = k\sqrt{density\times valveopening\times(P_1\times P_2)} \tag{5.6}$$

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:

$$valveopening = \left(\frac{\%\text{ valve open}}{100}\right)^2 \tag{5.7}$$

Nozzles Page

The **Nozzles** page contains information regarding the elevation and diameter of the nozzles. The Plant+ license is required to use the Nozzle features found on this page. Refer to **Section 1.5 - Plant+** for more information.

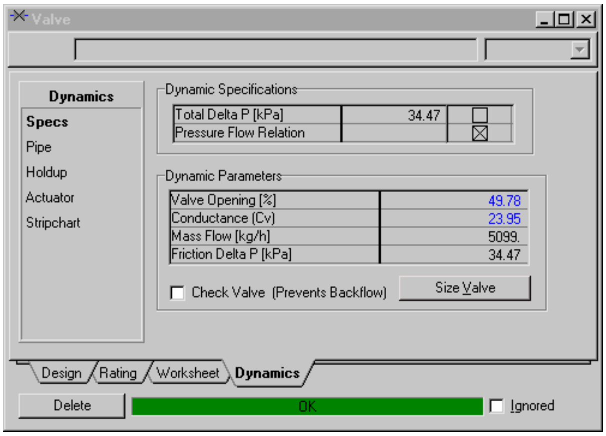
5.2.2 Dynamics Tab

The **Dynamics** tab provides access to four pages: the **Specs**, **Pipe**, **Holdup**, and **Actuator** pages.

Specs Page

The dynamic specifications and parameters of the valve can be specified on the **Sizing** page:

Figure 5.4



Dynamic Specifications

In dynamics mode, there are two possible dynamic specifications you can choose to characterize the valve operation.

If the **Total Delta P** box is checked, 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.

If the **Pressure Flow Relation** box is checked, 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** section lists the same stream and valve conditions required to size the valve as in the **Sizing Conditions** section. The Valve Opening % and the Conductance (Cv or k) are displayed and can be modified in the section. The conductance of the valve can be calculated by pressing the ***Size Valve*** button.

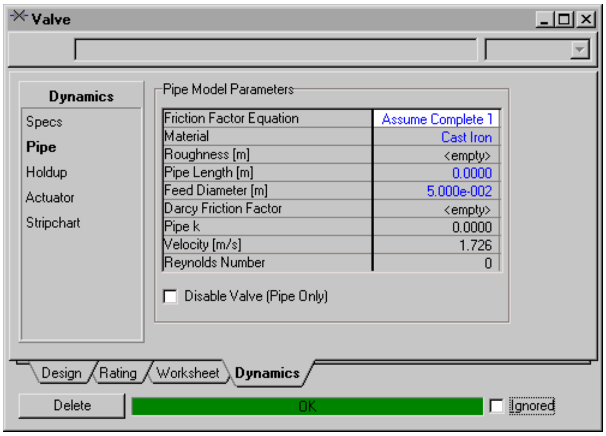
The **Check Valve** box can be checked if you do not want flow reversal to occur in the valve.

Pipe Page

Note that the pipe calculations for a valve are not rigorous for multiphase flow and the values will only be approximations.

The valve module supports a pipe contribution in the pressure flow equation. 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 modelled.

Figure 5.5



The following pipe modelling parameters are displayed in this section:

- Friction Factor Equation
- Material
- Roughness
- Pipe length
- Feed diameter
- Darcy friction factor
- Pipe k
- Velocity
- Reynolds number

The Friction Factor Equation option allows you to choose between two different equations: Assume Complete Turbulence (f is fixed) and Full-Range Churchill (covers all flow regimes). The default equation, Assume Complete Turbulence, is fast and simple. This method calculates the friction factor once and uses that value irrespective of the Reynolds number (this number will not be correct if the flow is laminar). 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 to low HYSYS will use a low limit of 10 for the Reynolds number.

HYSYS will suggest 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:

*Note: 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.*

$$\frac{1}{\sqrt{f_{friction}}} = 2.457 \ln\left(\frac{3.707D}{\varepsilon}\right) \quad (5.8)$$

$$f_{Darcy} = 8 \times f_{friction} \quad (5.9)$$

where: f_{Darcy} = *the Darcy friction factor*

D = *the pipe diameter*

ε = *the pipe roughness*

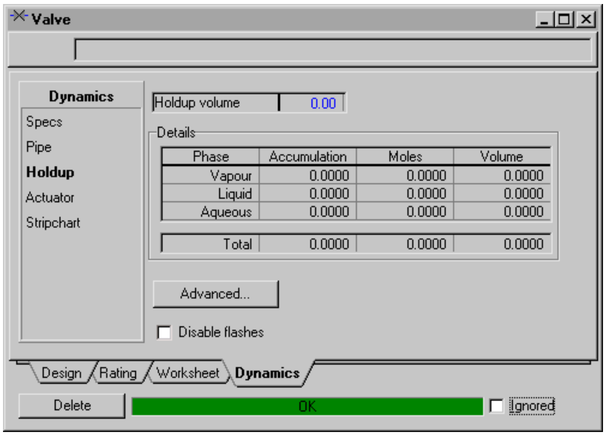
A pipe k-value is calculated from the Darcy friction factor and the pipe diameter. The pipe k value is incorporated into the general valve equation. Note that this pipe k is independent of the flow rate or pressure of the fluid in the valve.

Holdup Page

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. This page also allows you to disable any flashes that may occur in the valve.

The **Holdup** page contains information regarding the holdup properties, composition, and amount.

Figure 5.6



For each phase contained within the volume space of the unit operation, the following is specified:

Holdup Details	Description
Holdup Volume	The holdup volume can be specified in this cell.
Accumulation	The accumulation refers to the rate of change of material in the holdup for each phase.
Moles	These cells indicate the amount of material in the holdup for each phase.
Volume	These cells indicate the holdup volume of each phase.

Click the Advanced button to accesses a view that provides more detailed information about the holdup of that unit operation. Refer to Section 1.2.7 - Advanced Holdup Properties for more information.

Actuator Page

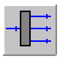
The **Actuator** page allows you to model valve dynamics in the valve operation. The Plant+ license is required to use the Actuator features found on this page. Refer to Section 1.5 - Plant+ for more information.

Stripchart Page

The **Stripchart** page allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the **Create Stripchart** button to display the stripchart.

5.3 Tee

The dynamic **TEE** operation splits one feed stream into multiple product streams with the same compositions and conditions as the feed stream. 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.



Tee Button

To install the **TEE** operation, press **F12** and choose **Tee** or select the ***Tee*** button in the Object Palette.

5.3.1 Rating Tab

Nozzles Page

The **Nozzles** page contains information regarding the elevation and diameter of the nozzles. The Plant+ license is required to use the Nozzle features found on this page. Refer to **Section 1.5 - Plant+** for more information.

It is strongly recommended that the elevation of the inlet and exit nozzles are equal for this unit operation. If you wish to model static head, the entire piece of equipment can be moved by modifying the **Base Elevation relative to Ground** cell.

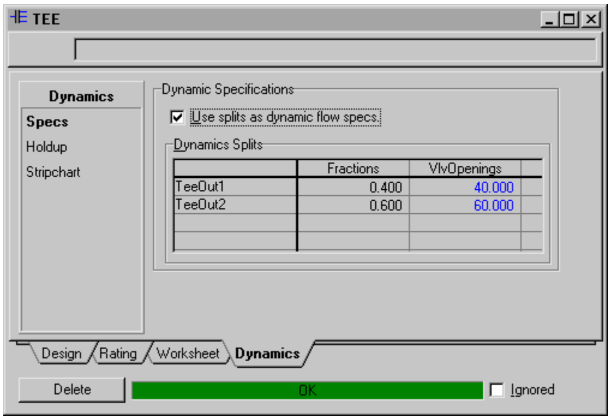
5.3.2 Dynamics Tab

The **Dynamics** tab provides access to two pages: the **Specs** and **Holdup** pages.

Specs Page

The dynamic specifications of the tee can be specified on the **Specs** page:

Figure 5.7



In **Dynamics** mode, there are two specifications you can choose to characterize the **TEE** operation.

If the **Use Splits as Dynamic Flow Specs** box is checked, 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** box is not checked, 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** box is not checked. It is strongly recommended that you uncheck the **Use Splits** box 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** box is unchecked. 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

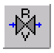
Each unit operation in HYSYS has the capacity to store material and energy. Typical tees in actual plants usually have significantly less holdup than other unit operations in a plant. Therefore, the volume of the **Tee** operation in HYSYS cannot be specified and is assumed to be zero. Since there is no holdup associated with the tee operation, the holdup's quantity and volume are shown as zero in the **Holdup** page.

Click the Advanced button to accesses a view that provides more detailed information about the holdup of that unit operation. Refer to **Section 1.2.7 - Advanced Holdup Properties** for more information.

Stripchart Page

The **Stripchart** page allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the **Create Stripchart** button to display the stripchart.

5.4 Relief Valve



Relief Valve Button

The **RELIEF VALVE** operation is used in many situations in which there has been excess pressure build up. Although it is available in **Steady State** mode, its purpose is to avert situations that occur in a dynamic environment.

To install the **RELIEF VALVE** operation, press **F12** and choose **Relief Valve** or select the **Relief Valve** button in the Object Palette.

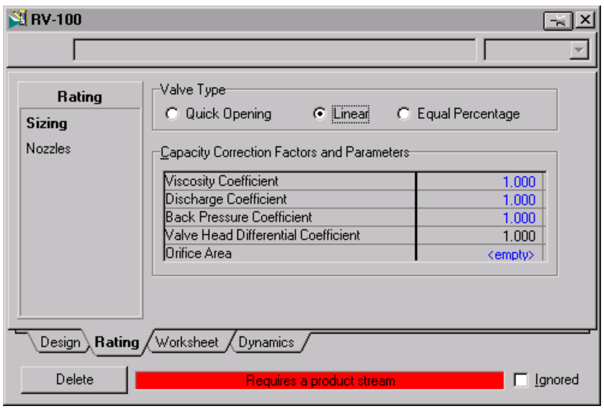
5.4.1 Ratings Tab

The **Rating** tab provides access to two pages: the **Sizing** page and the **Nozzles** page.

Sizing Page

On the **Sizing** page, you may specify the **Valve Type** and the **Capacity Correction Factors and Parameters**.

Figure 5.8



Valve Type

In HYSYS, you can specify three different valve characteristics for any relief valve in the simulation case.

Valve Type	Description
Linear	A relief valve with linear valve characteristics has a flow which is directly proportional to the valve % opening.
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.
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 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 will **NOT** be Choked:

$$\frac{P_2}{P_1} \geq \left[\frac{2}{K+1}\right]^{\frac{K}{K-1}} \tag{5.10}$$

where: P_2 = *Downstream Pressure*

K = *Ratio of Specific Heats*

For Choked vapour flow, the mass flowrate is given by the following relationship:

$$W = AK_LK_DK_B\left[\frac{P_1K}{V_1}\left[\frac{2}{K+1}\right]^{\frac{K+1}{K-1}}\right]^{\frac{1}{2}} \tag{5.11}$$

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:

$$W = AK_LK_D\left(\frac{P_1}{V_1}\left(\frac{2K}{K-1}\right)\left[\left(\frac{P_2}{P_1}\right)^{\frac{2}{K}} - \left(\frac{P_2}{P_1}\right)^{\frac{K+1}{K}}\right]\right)^{\frac{1}{2}} \tag{5.12}$$

Liquid Flow In Valve

Liquid Flow through the valve is calculated using the following equation:

$$W = AK_L K_D K_V[2(P_1 - P_2)\rho_1]^{\frac{1}{2}} \tag{5.13}$$

where: ρ_1 = *Density 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-supplied. 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:

$$K_L = \frac{L}{[a + (1 - a)L^4]^{1/2}} \tag{5.14}$$

Linear and decreasing-sensitivity valves have the following flow characteristics:

$$K_L = \frac{L^2}{[a + (1 - a)L^2]^{1/2}} \tag{5.15}$$

where:

$$a = \frac{\text{valve head differential a maximum flow}}{\text{valve head differential at zero flow}} \tag{5.16}$$

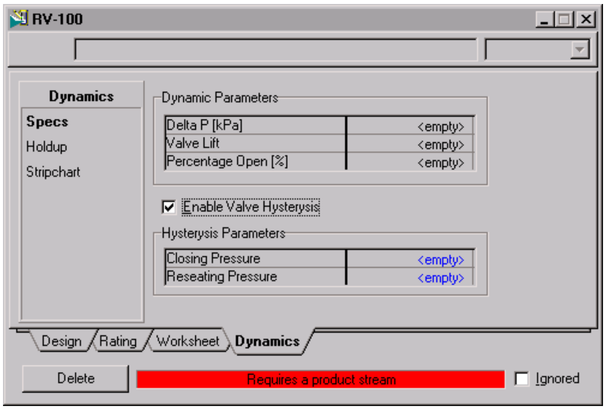
The valve head differential term allows for customization of the flow characteristics with respect to stem travel. Its value can range between 0 and 1.

Nozzles Page

The **Nozzles** page contains information regarding the elevation and diameter of the nozzles. The Plant+ license is required to use the Nozzle features found on this page. Refer to **Section 1.5 - Plant+** for more information.

5.4.2 Dynamics Tab

Figure 5.9



Specs Page

The **Specs** page consists of two groups: the **Dynamic Parameters** and **Hysterysis Parameters** groups.

Dynamic Parameters Group

The **Dynamic Parameters** 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: $$L = \left[\frac{P_1 - P_{OPEN}}{P_{FULL} - P_{OPEN}}\right] \quad (5.17)$$ *where:* P_1 = *upstream pressure* P_{OPEN} = *Set Pressure* P_{FULL} = *Full Open Pressure* If inlet pressure is decreasing: $$L = \left[\frac{P_1 - P_{RESEAT}}{P_{CLOSE} - P_{OPEN}}\right] \quad (5.18)$$ *where:* P_1 = *upstream pressure* P_{RESEAT} = *Reseating Pressure* P_{CLOSE} = *Closing Pressure*
Percentage Open	The Valve Lift in percentage.

Hysterysis Parameters Group

When the **Enable Valve Hysterysis** check box is activated, the **Hysterysis Parameters** group box becomes visible. This group contains two fields:

Field	Descriptions
Closing Pressure	Pressure at which the valve begins to close after reaching the full lift pressure (i.e. the value entered in the full pressure cell on the **Parameters** page of the **Design** tab).
Reseating Pressure	The pressure at which the valve *reseats* after discharge.

Holdup Page

For more information on the Holdup page this operations, consult the VALVE operations **Holdup Page**, Section 5.2.2 - Dynamics Tab.

Stripchart Page

The **Stripchart** page allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the **Create Stripchart** button to display the stripchart.

6 Rotating Equipment

6.1 Compressor/Expander

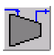
*All information in this section applies to the **COMPRESSOR** and **EXPANDER** unit operations, unless indicated otherwise*

The dynamics **COMPRESSOR** operation is used to increase the pressure of an inlet gas stream by providing energy in the form of shaft work to the system.

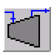
The dynamics **EXPANDER** operation is used to decrease an inlet gas stream's pressure. The expansion of the gas involves converting the internal energy of the gas to kinetic energy in the form of shaft work.

Some of the features included in the Dynamic **COMPRESSOR** and **EXPANDER** operations include:

- Dynamic Modelling of friction loss and inertia in the **COMPRESSOR**/**EXPANDER**.
- Dynamic Modelling which supports shutdown and startup behaviour.
- Multiple head and efficiency curves.
- Modelling of Stonewall and Surge conditions of the **COMPRESSOR**/**EXPANDER**.
- A dedicated surge controller which feature quick opening capabilities.
- Handling of phase changes that may occur in the unit operation (e.g. expanders producing liquid).
- Linking capabilities with other rotational equipment operating at the same speed with one total power.



Compressor Button



Expander Button

To install the **COMPRESSOR** or **EXPANDER** operation, press **F12** and choose the compressor or expander from the Unit Ops view or select the ***Compressor*** or ***Expander*** button in the Object Palette.

To ignore the **COMPRESSOR** or **EXPANDER**, select the **Ignore** check box. HYSYS will disregard the operation (and will not calculate the outlet stream) until you restore it to an active state by clearing the check box.

6.1.1 Theory

An essential concept associated with the **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:

$$W = F_1(MW)\left(\frac{n}{n-1}\right)CF\left(\frac{P_1}{\rho_1}\right) \times \left[\left(\frac{P_2}{P_1}\right)^{\left(\frac{n-1}{n}\right)} - 1\right] \tag{6.1}$$

where: n = *the volume exponent*

CF = *the 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:

$$n = \frac{\ln(P_2/P_1)}{\ln(\rho'_2/\rho_1)} \tag{6.2}$$

where: ρ'_2 = *density of the exit stream corresponding to the inlet entropy*

Polytropic power is calculated by defining the volume exponent as:

$$n = \frac{\ln(P_2/P_1)}{\ln(\rho_2/\rho_1)} \tag{6.3}$$

where: ρ_2 = *density of the exit stream*

The correction factor is calculated as:

$$CF = \frac{h'_2 - h_1}{\left(\frac{n}{n-1}\right)\left(\frac{P_2}{\rho'_2} - \frac{P_1}{\rho_1}\right)} \tag{6.4}$$

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.

6.1.2 Design Tab

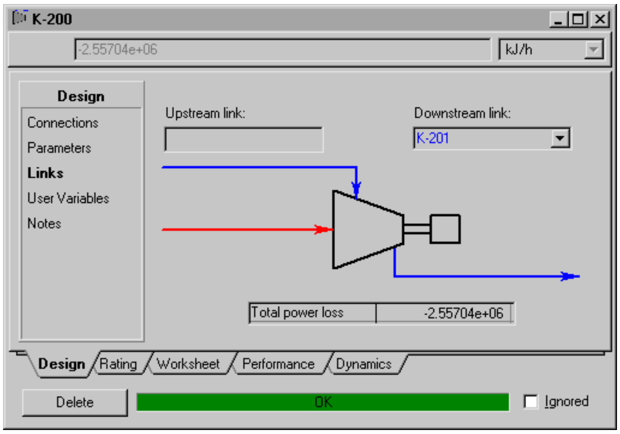
Links Page

Compressors and expanders modelled in HYSYS can have shafts which are physically connected. Linking compressors or expanders in HYSYS means:

- The speed of each linked unit operation is the same.
- The *Sum of Duties* of each linked compressor/expander + The *Total Power Loss* = 0. The Total Power Loss is located in the **Links** page of the **Design** tab.

The rotational equipment linker operates both in **Steady State** and **Dynamics** mode.

Figure 6.1



A list of available **COMPRESSORS/EXPANDERS** can be displayed by pressing the drop down button in the **Downstream Links** field or by pressing the **F2** key while the cursor is in the field. **Figure 6.1** shows a compressor K-200 which is linked to compressor K-201. It is not significant which order the compressors/expanders are linked. The notion of upstream and downstream links is arbitrary and determined by the user.

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 the Characteristic Curves** box selected in the **Specs** page of the **Dynamics** tab. In addition, the total power loss for the linked operations should be specified. In the case of linked compressors, it can be used to provide the total power by setting:

$$\textit{Total Power Input} = -\ \textit{Total Power Loss} \tag{6.5}$$

Usually, total power input to the linked compressors/expanders is calculated in a **Spreadsheet** operation and specified by the you in the **Total Power Loss** field.

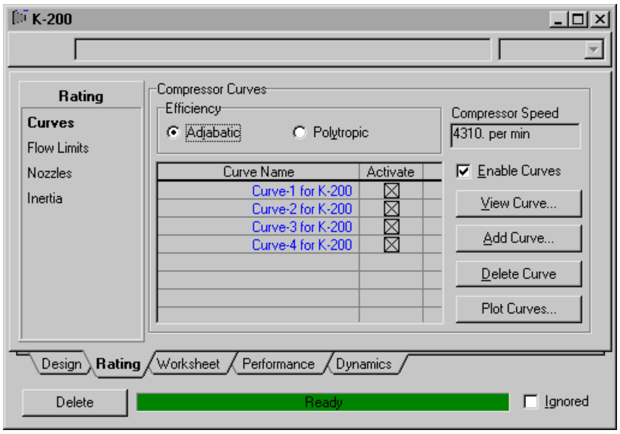
It is also possible to link an expander to a compressor and use the expander to generate kinetic energy to drive the compressor. If this option is chosen, the total power loss is typically specified as zero.

6.1.3 Ratings Tab

Curves Page

One or more compressor/expander curves may be specified on the **Curves** page. 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 compressor/expander. Multiple curves can be plotted to show the dependence of efficiency and head on the speed of compressors/expanders.

Figure 6.2

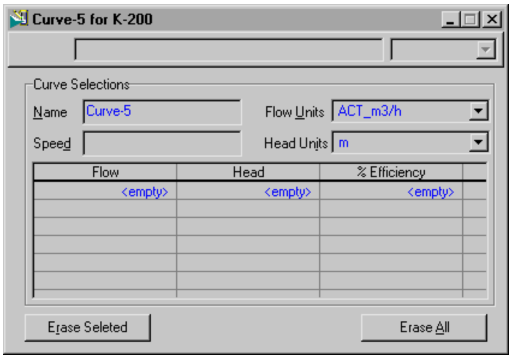


To supply data for a curve, follow this procedure:

1. Select the **Enable Curves** check box.
2. Choose an **Efficiency** Type radio button, either **Adiabatic** or **Polytropic**. This determines the basis of your input efficiency values.
3. Press the ***Add Curve*** button to access a **Curve** view.

4. On the **Curve** view, you may add **Flow**, **Head** and **%Efficiency** data points, as well as a **Speed** value for a single curve.

Figure 6.3



5. For each additional curve, repeat steps #3 and #4. The Efficiency radio button must be the same for input Curves.

*Each curve is named and has as associated **Activate** check box. You can turn individual curves on and off.*

HYSYS will use the curve(s) to determine the appropriate efficiency for your operational conditions. If you supply 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 ***View Curve***, ***Delete Curve*** and ***Plot Curves*** buttons will become available. By using these two buttons respectively, you can either access the **Curve** view to edit your input data or simply delete the highlighted curve from the simulation. Also, for each curve, an **Activate** check box will be present. To remove a specific curve from the calculations, you can deactivate its check box.

The **Curve** view is accessed via the ***Add Curve*** or ***View Curve*** button. You can supply the following data:

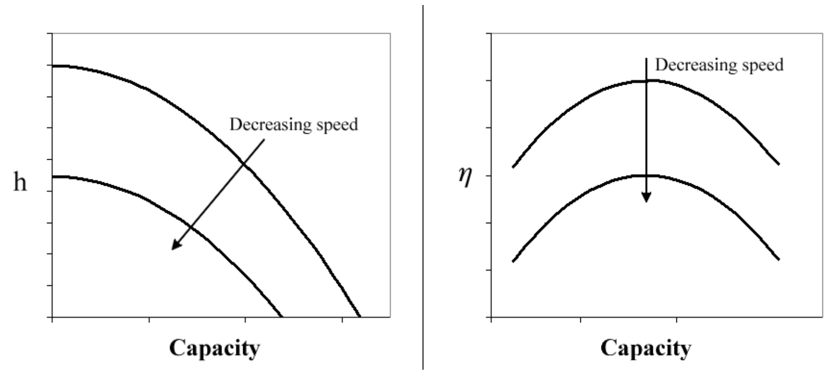
- **Name** - Name of this Curve.
- **Speed** - The rotation speed of the COMPRESSOR or EXPANDER. Unlike steady state, this specification is always required if you supply one curve or multiple curves.
- **Flow Units / Head Units** - Units for the Flow and Head (In Volume/Time and Length units, respectively).
- **Flow / Head / Efficiency** - Enter any number of data points for the Curve.

When you choose the **Erase selected** button, the current row (Flow, Head or Efficiency) will be deleted. Choose the **Erase All** button to delete all Flow, Head and Efficiency data for the Curve.

HYSYS can interpolate values for the efficiency and head of the compressor/expander for speeds that are not plotted

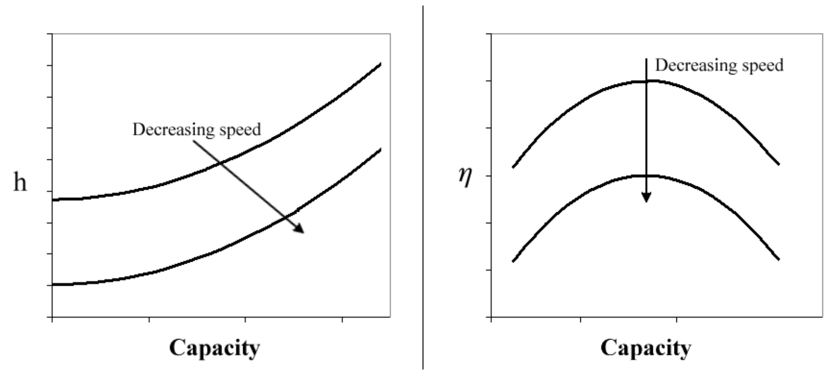
In order to run a stable and realistic dynamic model, HYSYS requires you to input reasonable curves. If compressors/expanders are linked, it is a good idea to ensure that the curves plotted for each unit operation span a common speed and capacity range. Typical curves are plotted in Figure 6.4 and Figure 6.5.

Figure 6.4



Typical Compressor Curve Shapes

Figure 6.5



Typical Expander Curve Shapes

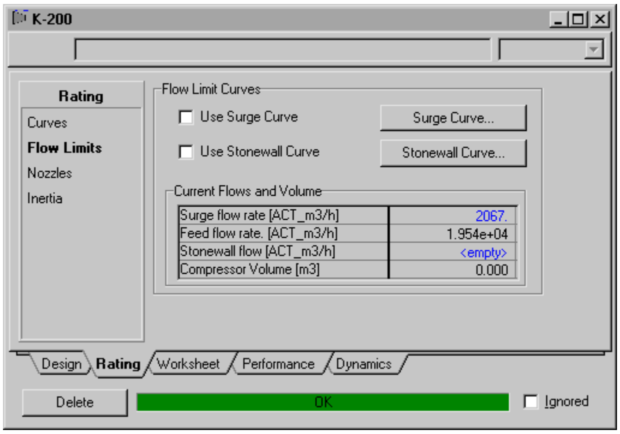
For an **EXPANDER**, the head is only zero when the speed and capacity are zero.

Note that a **COMPRESSOR** may 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 wish to pump a fluid near its critical point (where it becomes compressible), you may do so by representing the **PUMP** with a **COMPRESSOR**. The **COMPRESSOR** operation takes into account the compressibility of the liquid, thus performing a more rigorous calculation.

Flow Limits Page

There is a certain range that the dynamic **COMPRESSORS/ EXPANDERS** can operate in depending on its operating speed. The lower flow limit of a 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 compressor or expander by plotting surge and stonewall curves.

Figure 6.6

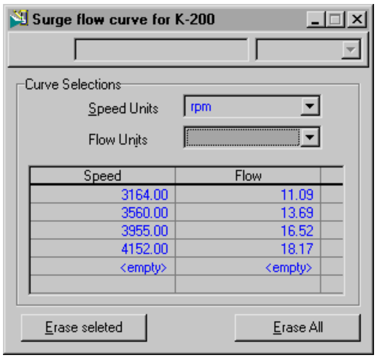


From the **Flow Limits** page, it is possible to add **Surge** or **Stonewall** curves for the compressor. To add or edit a **Surge** curve, follow this procedure:

1. Press the **Surge Curve** button.
2. Select the required units for **Speed** and **Flow**.
3. Add **Speed** and **Flow** data points for the curve.

4. To remove a certain data points, highlight either the **Speed** or **Flow** cells and press the **Erase selected** button.
5. To remove all the data points, press the **Erase All** button.

Figure 6.7



The procedure for adding or editing **Stonewall** curves is identical.

When a dynamic compressor reaches its stonewall limit, HYSYS will fix the flow at that compressor speed. When a compressor reaches the surge limit, the flow reverses and cycles continuously causing damage to the compressor. This phenomenon is modelled in HYSYS by causing the flow rate through the compressor to fluctuate randomly below the surge flow.

Nozzles Page

The **Nozzles** page contains information regarding the elevation and diameter of the nozzles. The Plant+ license is required to use the Nozzle features found on this page. Refer to **Section 1.5 - Plant+** for more information.

For a Compressor/Expander unit operation it is strongly recommended that the elevation of the inlet and exit nozzles are equal. If you wish to model static head, the entire piece of equipment can be moved by modifying the **Base Elevation relative to Ground** cell.

Inertia Page

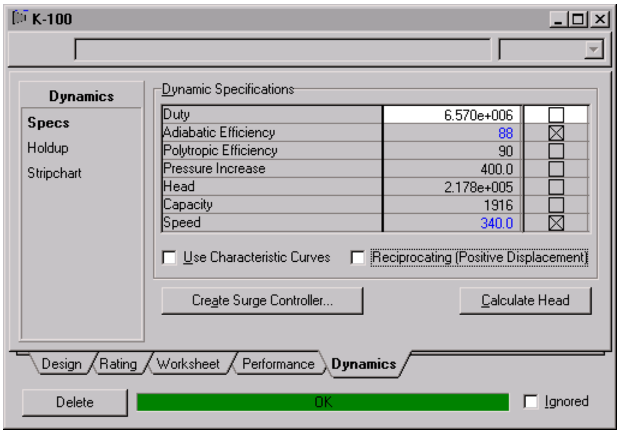
The inertia modelling parameters and the frictional loss associated with the impeller in the Compressor can be specified on the **Inertia** page. The Plant+ license is required to use the Inertia features found on this page. Refer to **Section 1.5 - Plant+** for more information.

6.1.4 Dynamics Tab

Specs Page

The dynamic specifications of the **COMPRESSOR/EXPANDER** can be specified on the **Specs** page:

Figure 6.8



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** box should not be selected if the inlet and exit stream pressures are specified.
- The **Speed** box should not be selected if the **Use Characteristic Curves** check box is not selected.

The possible dynamic specifications are as follows:

Duty

The duty is defined, in the case of the **COMPRESSOR** operation, as the power required to rotate the shaft and provide energy to the fluid. The duty has three components:

$$\text{Duty} = \text{Power imparted to the fluid} + \text{Power required to change the rotational speed of the shaft} + \text{Power lost due to mechanical friction loss} \quad (6.6)$$

The duty in a 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 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:

$$\eta = \frac{W(to\ system)}{F_1(MW)(h_2 - h_1)} \quad (6.7)$$

For a dynamic expander, the efficiency is defined as:

$$\eta = \frac{F_1(MW)(h_1 - h_2)}{W(from\ system)} \quad (6.8)$$

If a polytropic efficiency definition is required, the polytropic work should be provided in Equation (6.7) or Equation (6.8). If an adiabatic efficiency definition is required, the isentropic work should be provided.

Note that 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 5.1.1 - Theory in the **Steady State Modeling** manual. This is true since the actual work required by a steady state compressor is the same as the energy imparted to the fluid.

If compressor or expander curves are provided in the **Curves** page of the **Ratings** tab, the isentropic or polytropic efficiency can be interpolated from the flow of gas and the speed of the compressor/ expander.

Pressure Increase

A **Pressure Increase** specification can be selected if the pressure drop across the 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:

$$W = (MW)F_1(CF)gh \tag{6.9}$$

where: W = *the isentropic or polytropic power*

MW = *the molecular weight of the gas*

CF = *the correction factor*

F_1 = *the molar flow rate of the inlet gas stream*

g = *gravity acceleration*

If **COMPRESSOR** or **EXPANDER** curves are provided in the **Curves** page of the **Ratings** tab, the isentropic or polytropic head can be interpolated from the flow of gas and the speed of the compressor/expander.

Capacity

The capacity is defined as the actual volumetric flow rate entering the **COMPRESSOR/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 compressor or being driven by the expander can be specified.

Use Characteristic Curves

Select the **Use Characteristic Curves** check box if you wish to use the curve(s) specified in the **Curves** page of the **Ratings** tab. If a single curve is specified in a dynamics compressor, the speed of the compressor is not automatically set to the speed of the curve unlike the steady state **COMPRESSOR/EXPANDER** unit operation. A different speed may be specified and HYSYS will extrapolate values for head and efficiency.

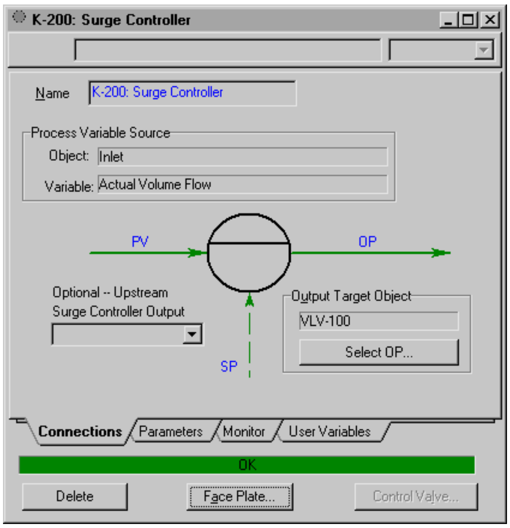
Reciprocating (Positive DIsplacement)

Select the **Reciprocating (Positive DIsplacement)** check box if you wish to use a reciprocating compressor instead of a centrifugal compressor. Only available with the **Compressor** unit operation, refer to **Section 6.2 - Reciprocating Compressor** for more information.

Surge Control

The **Create Surge Control** button opens a **Surge Control** view which is owned by the compressor. If you decide to delete the compressor, the surge controller associated with the compressor is deleted as well. Surge control also works exclusively with compressor and expander unit operations.

Figure 6.9



As mentioned, a compressor will surge if its capacity falls below the surge limit. The surge controller determines a minimum volumetric flowrate that the compressor should operate at without surging. This is called the surge flow. The surge controller then attempts to control the flow to the compressor at some percent above the surge flow, typically 10%. The surge controller essentially acts like **PID Controller** operations. The control algorithms used to prevent compressors from surging are extensions of the PID algorithm.There are two major differences which distinguish a surge controller and a regular controller:

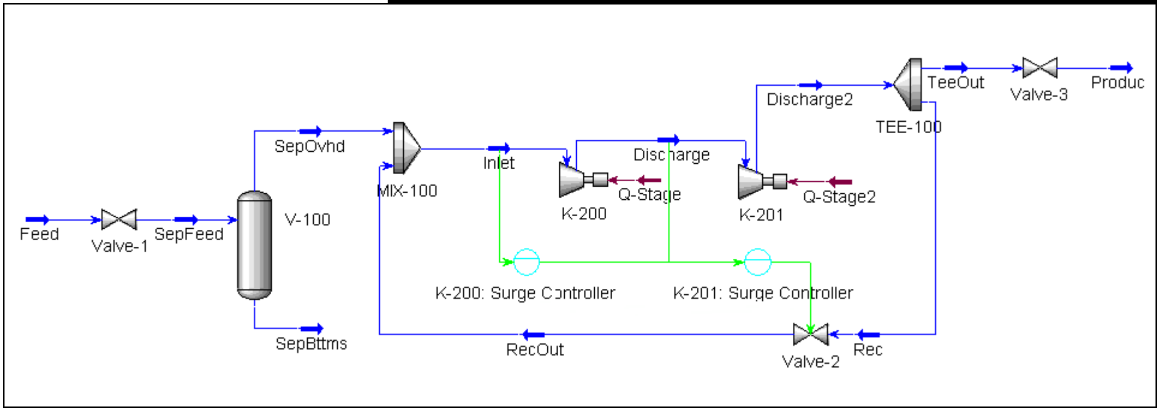
- The setpoint of the surge controller is calculated and not set.
- More aggressive action is taken by the surge controller if the compressor is close to surging.

Connections Tab

The **Connections Tab** is very similar to a PID controller's connection tab. For more information on the individual parameters which make up the **Connections Tab**, see **Chapter 10 - Logical Operations**. The inlet volumetric flow to the compressor is automatically defaulted as the process variable, PV, to be measured. You must choose a **CONTROL VALVE** operation as an operating variable, OP, which has a direct effect on the inlet flow to the 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 will ensure that the output signal of the 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 6.10



As shown in **Figure 6.10**, 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

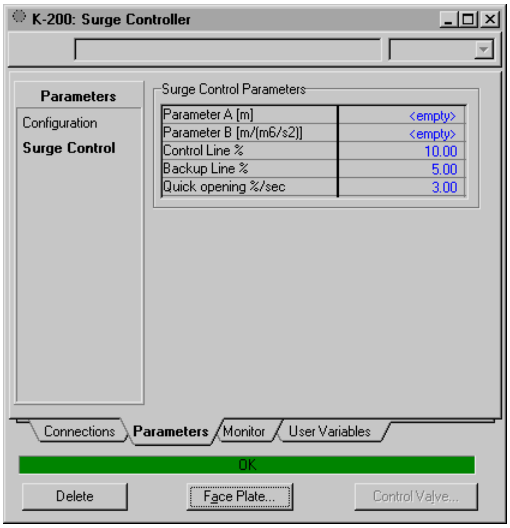
Configuration Page

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. For more information on the individual fields in the **Configuration** page, see **Section 10.1 - PID Controller**.

Surge Control Page

Various surge control parameters can be specified in the **Surge Control** page.

Figure 6.11



A head versus quadratic flow expression relates the surge flow to the head of the compressor. The surge flow, F_S, is calculated as follows:

$$F_S(m^3/s) = \sqrt{\frac{h(m) - A}{B}} \qquad (6.10)$$

You can enter surge flow parameters A and B in order to characterize the relationship between 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 control line is 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 (%)	The backup line is 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 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 compressor rises above the backup line.

Holdup Page

Typical compressors and expanders in actual plants usually have significantly less holdup than most other unit operations in a plant. Therefore, the volume of the **Compressor/Expander** operation in HYSYS cannot be specified and is assumed to be zero on the **Holdup** page.

Stripchart Page

The **Stripchart** page allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the **Create Stripchart** button to display the stripchart.

6.1.5 Linked Compressor Example

Two compressors will be built and physically linked in **Dynamics** mode. The compressors will first be built in steady state mode and then modified slightly in Dynamics mode. Consider the LP gas from an Oil Production facility as a **FEED** to a compressor station.

Create a fluid package using the **Sour PR** property package and add the following components: H2O, H2S, CO2, C1, C2, C3, i-C4, and n-C4.

Property Package	Components
Sour PR	H2O, H2S, CO2, C1, C2, C3, i-C4, n-C4

1. Install the stream **FEED** and define it as follows:

Name	Feed
Temperature [°C]	69.000
Pressure [kPa]	120.00
Molar Flow [kgmole/hr]	500.00
Comp Mole Frac [H2O]	0.2376
Comp Mole Frac [H2S]	0.0727
Comp Mole Frac [CO2]	0.0607
Comp Mole Frac [C1]	0.0430
Comp Mole Frac [C2]	0.1072
Comp Mole Frac [C3]	0.2522
Comp Mole Frac [i-C4]	0.0765
Comp Mole Frac [n-C4]	0.1502

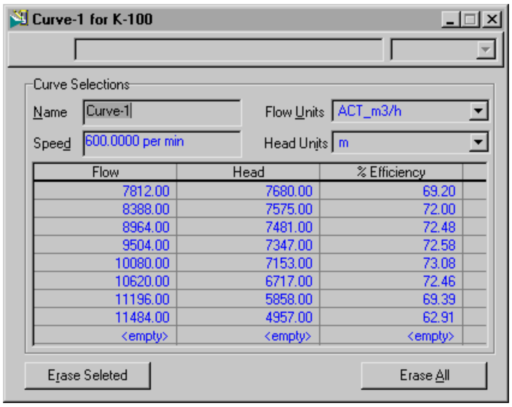
2. Install a **COMPRESSOR** unit operation with the following information:

Tab [Page]	Input Area	Entry
Design [Connections]	Name	K-100
	Inlet	Feed
	Outlet	Discharge
	Energy	Comp Duty

3. On the **Parameters** page, delete the default **Adiabatic Efficiency** value. Both the Adiabatic and Polytropic efficiencies should read <*empty*> to avoid a consistency error or a compressor overspecification.
4. Switch to the **Rating** tab, **Curves** page.
5. Ensure that the **Adiabatic** radio button in the **Efficiency** group is selected.
6. Check the **Enable Curves** box, and press the ***Add Curve*** button.

7. Complete the curve as shown in Figure 6.12. Since this compressor will be eventually be operating in Dynamics mode, you must enter the speed of the compressor.

Figure 6.12



8. Close the **Curve** view and check the **Activate** box for the curve that was just created.
9. Install another **COMPRESSOR** unit operation with the following information:

Tab [Page]	Input Area	Entry
Design [Connections]	Name	K-101
	Inlet	Discharge
	Outlet	Out
	Energy	Comp Duty2

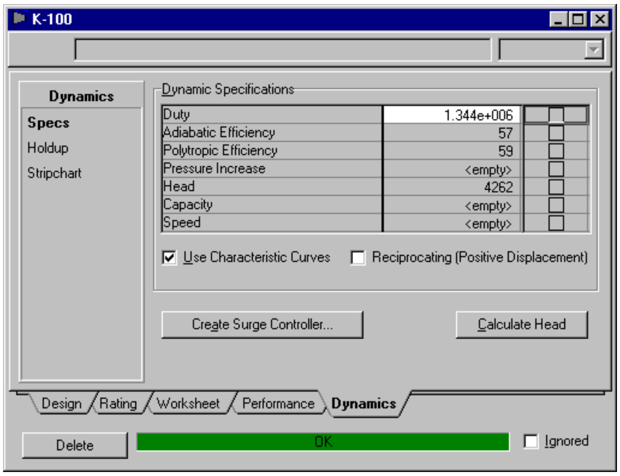
Dynamics Mode Button

Dynamics Assistant Button

10. Specify the second compressor like the first by repeating steps #3 to#8.
11. Switch to **Dynamics** mode.
12. The **Dynamics Assistant** will specify the compressor operations with a polytropic efficiency and a duty. However, the compressor curves are required by HYSYS in order to link the compressors. Deactivate the Polytropic efficiency and Duty specification and

activate the **Use Characteristic Curves** specification in the **Specs** page of the **Dynamics** tab. Change the specifications for both compressors.

Figure 6.13



Downstream link drop down list

13. On the **Links** page of the **Design** tab in the K-100 compressor, select the **K-101** compressor from the Downstream Link drop down list.
14. In the **Total Power Loss** cell on the **Links** page of the **Design** tab, specify a value of -3.0 x 10^6 kJ/h.
15. Run the integrator until the compressor system reaches steady state conditions (approximately 2 minutes, integration time).
16. The conditions of stream **OUT** at steady state conditions are displayed as follows:

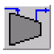
Name	Out
Vapour Fraction	1.0000
Temperature [°C]	625.4958
Pressure [kPa]	484.1
Molar Flow [kgmole/hr]	55.21
Mass Flow [kg/hr]	2077.8
Liq Vol Flow [m3/hr]	3.691
Heat Flow [kJ/hr]	-5.1635e+06

6.2 Reciprocating Compressor

In **Section 6.1 - Compressor/Expander**, a dynamic centrifugal compressor unit operation is presented. The following section will discuss the reciprocating compressor in dynamics. The reciprocating compressor is just another type of compressor used for applications where higher discharge pressure and lower flow are needed. It is known as positive displacement type. The reciprocating compressor has a constant volume and variable head characteristics, as compared to centrifugal compressor that has a constant head and variable volume.

In HYSYS, centrifugal and reciprocating compressors are accessed via the dame compressor unit operation. However, the solution methods differ slightly as a reciprocating compressor does not require a compressor curve and required geometry data. The present capability of reciprocating compressors in HYSYS is focused on a **single stage compressor** with single or double acting piston. Some of the features included in the Dynamic Reciprocating compressor unit operation include:

- Dynamic modeling of friction loss and inertia.
- Dynamic modeling which supports shutdown and startup and shutdown behaviour.
- Dynamic modeling of the cylinder loading.
- Linking capabilities with other rotational equipment operating at the same speed with one total power.



Compressor Button

To install the compressor operation, press **F12** and select compressor from the unit Ops view or select the Compressor button in Object Palette, **F4**. To toggle from a centrifugal compressor to a reciprocating compressor, the Reciprocating (Positive Displacement) check-box on **Specs** page of the **Dynamics** tab must be checked.

To ignore the Reciprocating compressor during calculation, select the **Ignored** check box. HYSYS will completely disregard the operation until you restore it to an active state by clearing the check box.

6.2.1 Theory

A reciprocating compressor is a compressor which transform the mechanical energy in the form of shaft work to boost the pressure of the gas contained in the cylinder. The fundamental calculation of this unit operation is the power calculation and associated equations to evaluate the performance of the unit. Volumetric efficiency, clearance volume, positive displacement volume and duty required are some of the performance evaluation parameters.

Cylinder clearance, C is given as

$$C = \frac{\text{Sum of all clearance volume for all cylinders}}{PD} \tag{6.11}$$

where:

PD is the positive displacement volume.

The sum of all clearance volume for all cylinders includes both fixed and variable volume. C is normally expressed in 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 may be expressed:

For single-acting piston compressing on the outer end only:

$$PD = \frac{\pi \cdot D^2 \cdot \text{stroke}}{4} \tag{6.12}$$

For single-acting piston compressing on the crank end only:

$$PD = \frac{\pi \cdot (D^2 - d^2) \cdot \text{stroke}}{4} \tag{6.13}$$

For double-acting piston (other than tail rod type):

$$PD = \frac{\pi \cdot (2D^2 - d^2) \cdot \text{stroke}}{4} \tag{6.14}$$

For double-acting piston (tail rod type):

$$PD = \frac{\pi \cdot (2D^2 - 2d^2) \cdot \text{stroke}}{4} \tag{6.15}$$

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 parameter 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:

$$VE = \left[(1-L) - C\left[\frac{Z_s}{Z_d}\left(\frac{P_d}{P_s}\right)^{\frac{1}{k}} - 1\right]\right] \tag{6.16}$$

where:

P_d *- Discharge pressure.*

P_s *- Suction pressure.*

L - the 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.*

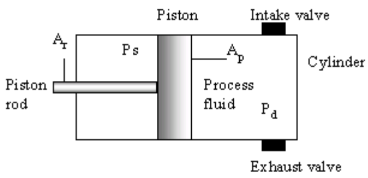
C - Clearance volume

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 6.14



Single Stage - Single Acting Piston

It may be calculated as follows:

Load in compression, L_c

$$L_c = P_d A_p - P_s(A_p - A_r) \tag{6.17}$$

Load in tension, L_t

$$L_t = P_d(A_p - A_r) - P_s A_p \tag{6.18}$$

Maximum Pressure

The maximum pressure that the reciprocating compressor can achieve is:

$$P_{max} = P_s \cdot PR_{max} \tag{6.19}$$

where the maximum discharge pressure ratio, PR_{max} is calculated from:

$$PR_{max} = \left[\frac{Z_d}{Z_s \cdot C}(1 - L - VE + C)\right]^k \tag{6.20}$$

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 will be zero or larger then zero (but never negative). The molar flow is then equal to:

$$\left[\left(1 - \frac{L}{100}\right) - C\left[\frac{Z_s}{Z_d}\left(\frac{P_d}{P_s}\right)^{\frac{1}{k}} - 1\right]\right]\left[\frac{\frac{N}{60} \cdot PD \cdot \rho}{MW}\right] \tag{6.21}$$

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 will be governed by a typical pressure flow relationship and the user can specify the resistance in zero speed flow resistance, k_{zero_speed}. The flow equation is as follows:

$$\text{MolarFlow, F} = k_{\text{zero speed}} \cdot \sqrt{\rho \cdot \Delta P_{\text{friction}}} \tag{6.22}$$

where:

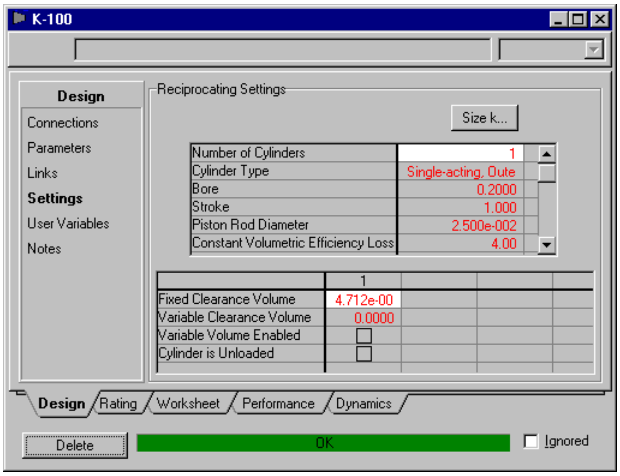
$P_{friction}$ = *frictional pressure drop across the compressor*

6.2.2 Design Tab

Settings Page

The **Settings** page is used to size your reciprocating compressor. This 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.

Figure 6.15



A reciprocating compressor does not require a characteristic curve, however the following compressor geometry information is required.

- Number of Cylinders
- Cylinder Type
- Bore - diameter of the cylinder
- Stroke - 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
- Volume Efficiency

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.

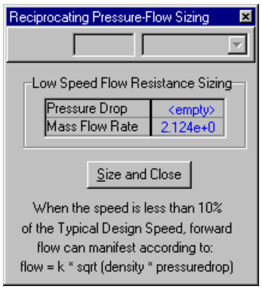
- Fixed Clearance Volume
- Variable Clearance Volume
- Variable Volume Enabled
- Cylinder is Unloaded - dynamics only

If the Variable Volume Enabled checkbox is checked you need to specify a Variable Clearance Volume. The variable volume is used when additional clearance volume (external) is intentionally added to reduce cylinder capacity.

If the Cylinder is Unloaded checkbox is checked the total displacement volume is not considered, and is essentially zero.

The **Size k** button allows you to specify a Pressure Drop and Mass Flow Rate that will be used to calculate the Zero Speed Flow Resistance of the reciprocating compressor.

Figure 6.16



6.2.3 Ratings Tab

The **Rating** tab contains two pages: **Nozzles** and **Inertia** page.

Nozzle Page

The **Nozzle** page is identical to the centrifugal compressor, refer to Section - Nozzles Page.

Inertia Page

The **Inertia** page is identical to the centrifugal compressor, refer to Section - Inertia Page.

6.2.4 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, Properties,** and **Composition** pages contain selected information from the corresponding pages of the **Worksheet** tab for the stream property view. The **PF Specs** page contains a summary of the stream property view **Dynamics** tab.

6.2.5 Dynamics Tab

The **Dynamics** tab is identical to the centrifugal compressor, refer to Section 6.1.4 - Dynamics Tab. However when using a reciprocating compressor you can not use the Characteristic Curves specification or created a Surge Controller.

6.2.6 Reciprocating Compressor Example

A typical, partial datasheet for a reciprocating compressor K-100 is presented as follows:

K-100 Reciprocating Compressor Datasheet				
General Data Per Machine				
Speed (rpm)		340		
Volumetric Efficiency (%)		90		
Stroke (in)		15		
Driver flywheel effect (lb-ft^2)		115000		
Number of Cylinder		2	(1 Stage)	
Weight of driver (lb)		4000		
Specific Data Cylinder No:		1	2	
Bore (in)		18	18	
Piston Rod diameter (in)		4	4	
Reciprocating weight		2000	2000	
Number of unloading mode		2	2	
Head load (Full, None)		Full	Full	
Crank load (Full, None)		Full	Full	

P suction, psia	HEAD	230	230	100% Load
P discharge, psia		630	630	
Clearance (%)		12.7	12.7	
P suction, psia	CRANK	230	230	
P discharge, psia		630	630	
Clearance (%)		12.8	12.8	
Flow, MMSCFD				43
Head load (Full, None)		None	Full	
Crank load (Full, None)		Full	Full	
P suction, psia	HEAD	230	230	75% Load
P discharge, psia		230	630	
Clearance (%)			12.7	
P suction, psia	CRANK	230	230	
P discharge, psia		630	630	
Clearance (%)		12.8	12.8	
Flow, MMSCFD				32.3
Head load (Full, None)		None	Full	
Crank load (Full, None)		Full	None	
P suction, psia	HEAD	230	230	50% Load
P discharge, psia		230	630	
Clearance (%)			12.7	
P suction, psia	CRANK	230	230	
P discharge, psia		630	230	
Clearance (%)		12.8		
Flow, MMSCFD				21.2
Head load (Full, None)		None	None	
Crank load (Full, None)		Full	None	
P suction, psia	HEAD	230	230	25% Load
P discharge, psia		230	230	
Clearance (%)				
P suction, psia	CRANK	230	230	
P discharge, psia		630	230	
Clearance (%)		12.8		
Flow, MMSCFD				10.7

Refer to **Section 5.2.7 - Reciprocating Compressor Example**, in the Steady State Modelling Guide, for the steps to setup the compressor in steady state for 100% load. Once you have setup the compressor setup continue on with the following steps.

1. On the Inertia page, input the driver weight, which is the mass of the impeller and the rotating shaft, into the Mass field and vary the

Radius of Gyration until a desirable Flywheel effect is achieved. This is just an estimate only because the inertia and the Flywheel effect is not the same. When the compressor changes it's speed, like during shutdown or startup, the inertia effect will control the time required to perform the speed change. A flywheel effect, which is small, eventually allows the compressor driver to reach its' speed faster.

2. Set the following dynamic specification on the **Specs** page of the **Dynamics** tab.
    - Disable the Duty specification.
    - Disable the Adiabatic Efficiency specification.
    - Enable the Polytropic Efficiency specification.
    - Enable the Speed specification and set the spec to 340 rpm.
3. Enter Dynamics mode by clicking the **Dynamics Mode** button in the toolbar. Click **No** when asked if you would like to use the Dynamic Assistant.

*Use the **Workbook** tab to view the results of the load changes.*

4. Start the Integrator and simulate the various load changes by checking the appropriate **Cylinder is Unloaded** checkbox on the **Settings** page of the **Design** tab. This steps is necessary to ensure the compressor is performing according to the design.

6.3 Pump

The dynamics **PUMP** operation is used to increase the pressure of an inlet liquid stream by providing energy in the form of shaft work to the system.

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 included in the Dynamic **PUMP** include:

- Dynamic Modelling of friction loss and inertia in the **COMPRESSOR/EXPANDER**.
- Dynamic Modelling which supports shutdown and startup behaviour.
- Multiple head and efficiency curves.
- Modelling 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.

The **On** switch in the **PUMP** operation view activates and deactivates the pump. If this box is selected, the pump is on and works normally. If this box is not selected, HYSYS sets the power to the pump to zero. In order to switch off the pump, the speed and/or power to the pump must be specified.

Pump Button

To install the **PUMP** operation, press **F12** and choose the **PUMP** from the **Unit Ops** view or select the **PUMP** button in the Object Palette.

To ignore the **PUMP**, select the **Ignore** check box. HYSYS will completely disregard the operation (and will not calculate the outlet stream) until you restore it to an active state by clearing the check box.

6.3.1 Theory

The ideal power required, W, to increase the pressure of an incompressible fluid is:

$$W = \frac{(P_2 - P_1)F(MW)}{\rho} \tag{6.23}$$

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 = *the molecular weight of the fluid*

Note that for a pump, it is assumed that the entering liquid stream is incompressible. Therefore, the ideal work defined in **Equation (6.23)** 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.

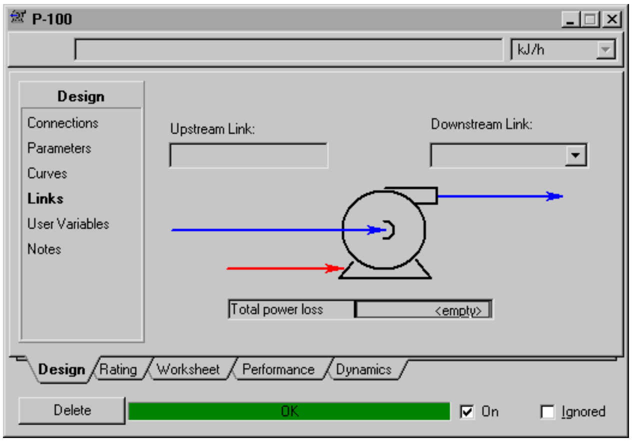
Incompressibility is the usual assumption for liquids well removed from the critical point, and the standard pump equation provided in **Equation (6.23)** is generally accepted for calculating the power requirement. However, if you wish to perform a more rigorous calculation for pumping a compressible liquid (i.e. one near the critical point), you should install a **COMPRESSOR** operation instead of a **PUMP**.

6.3.2 Design Tab

Links Page

In HYSYS, pumps can have shafts which are physically connected. The rotational equipment linker operates both in steady state and dynamics mode.

Figure 6.17



A list of available pumps can be displayed by pressing the drop down button in the **Downstream Links** field or by pressing the **F2** key while the cursor is in the field. It is not significant which order the pumps are linked. The notion of upstream and downstream links is arbitrary and determined by you, the user.

Linked **PUMP** operations require curves. To fully define a set of linked pumps, you must check the **Use the Characteristic Curves** box for each of the linked pumps in the **Specs** page of the **Dynamics** tab. One additional dynamic specification is required for the set. Usually, the total power loss from the linked operations is specified. For a series of linked pumps, it is desired to input a total power:

$$\text{Total Power Input} = -\ \text{Total Power Loss} \tag{6.24}$$

Total power loss or input to the linked pump operations can be specified by you in the **Total Power Loss** field.

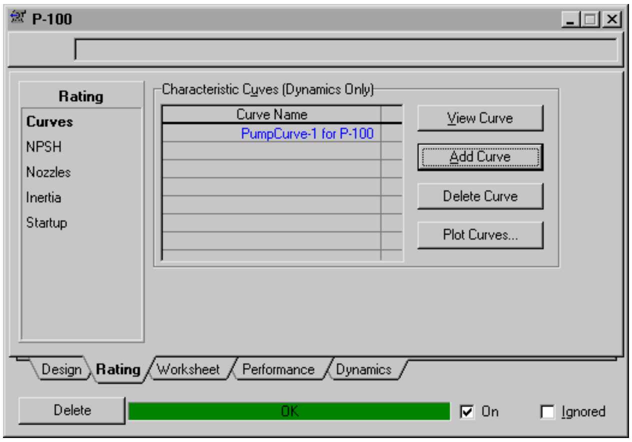
It is possible to link a pump to a compressor and use the pump as a turbine to generate kinetic energy to drive the compressor. If this option is chosen, the total power loss is typically specified as zero.

6.3.3 Ratings Tab

Curves Page

One or more pump curves may be specified on the **Curves** page. The curves used on this page work exclusively in **Dynamic** mode. Pump curves for steady state operation are specified in the **Curves** page of the **Design** tab. The efficiency and head for a specified speed can be plotted against the capacity of the pump. Multiple curves can be plotted to show the dependence of efficiency and head on the speed of pump.

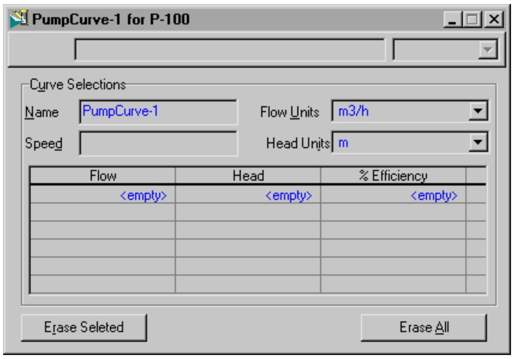
Figure 6.18



To supply data for a curve, follow the procedure outlined below:

1. Press the ***Add Curve*** button to access a **Curve** view.
2. On the **Curve** view, you may add **Flow**, **Head** and **%Efficiency** data points, as well as a **Speed** value for a single curve.

Figure 6.19



3. For each additional curve, repeat steps #1 and #2.

HYSYS will use the curve(s) to determine the appropriate efficiency for your operational conditions. If you supply 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 ***View Curve***, ***Delete Curve*** and ***Plot Curves*** buttons will become available. By using these three buttons respectively, you can either access the Curve view to edit your input data or simply delete the highlighted curve from the simulation.

The **Curve** view is accessed via the **Add Curve** or **View Curve** button. You can supply the following data:

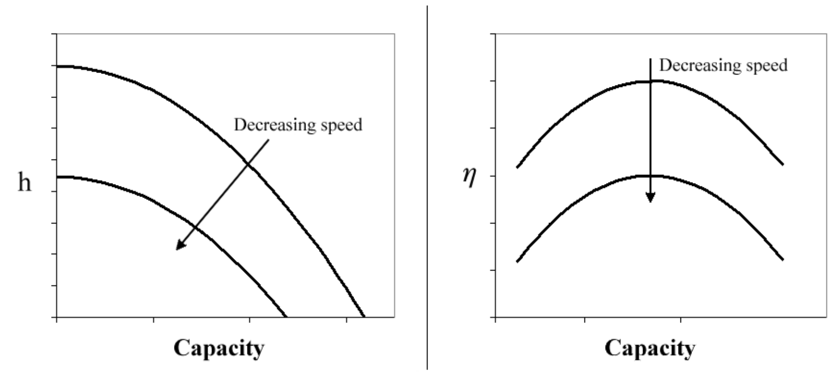
- **Name** - Name of this Curve.
- **Speed** - The rotation speed of the Pump. This is optional if you supply only one curve.
- **Flow Units / Head Units** - Units for the Flow and Head (In Volume/Time and Length units, respectively)
- **Flow / Head / Efficiency** - Enter any number of data points for the Curve.

When you choose the ***Erase Selected*** button, the current row (Flow, Head or Efficiency) will be deleted. Choose the ***Erase All*** button to delete all Flow, Head and Efficiency data for the curve.

HYSYS can interpolate values for the efficiency and head of the compressor/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/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 **Figure 6.20**.

Figure 6.20



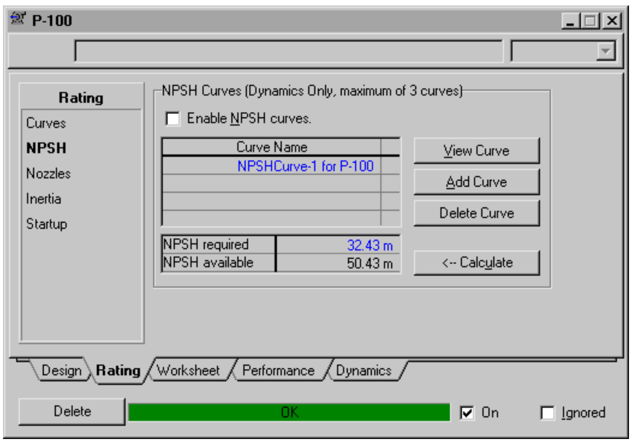
Typical Pump Curve Shapes

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:

Figure 6.21



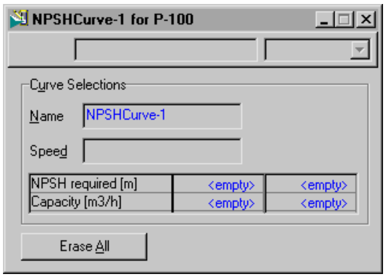
To add or edit a **NPSH** curve from the **NPSH** page, follow this procedure:

1. Select the **Enable NPSH curves** check box.
2. Press the ***Add Curve*** button to access the curve view.
3. Add a **Speed** for each curve.
4. Enter a capacity and NPSH for two points on the curve. Only two points are required for NPSH curves since:

$$\log(NPSH_{required}) \propto \log(capacity) \tag{6.25}$$

5. To remove all the data points, press the ***Erase All*** button.

Figure 6.22



6. For each additional curve, repeat steps #2 to #6.

The $NPSH_{required}$ value can either be taken from the NPSH curves or specified directly in the **NPSH required** cell. To directly specify the $NPSH_{required}$, you must first deselect the **Enable NPSH curves** check box.

$NPSH_{available}$ can be explicitly calculated from the flowsheet conditions by pressing the **Calculate Head** button. The $NPSH_{available}$ is calculated as follows:

$$NPSH_{available} = \frac{P_1 - P_{vap}}{\rho g} + \left(\frac{V_1^{\,2}}{2g}\right) \tag{6.26}$$

where: P_1 = *inlet stream pressure to the pump*

P_{vap} = *vapour pressure of the inlet stream*

ρ = *the density of the fluid*

V_1 = *the velocity of the inlet stream*

g = *the gravity constant*

To prevent pump cavitation the $NPSH_{available}$ must be above the $NPSH_{required}$. If a pump cavitates in HYSYS, it will be modelled by scaling the density of the fluid, ρ, randomly between zero and one.

Nozzles Page

The **Nozzles** page contains information regarding the elevation and diameter of the nozzles. The Plant+ license is required to use the Nozzle features found on this page. Refer to **Section 1.5 - Plant+** for more information.

For a Pump unit operation it is strongly recommended that the elevation of the inlet and exit nozzles are equal. If you wish to model static head, the entire piece of equipment can be moved by modifying the **Base Elevation relative to Ground** cell.

Inertia Page

The inertia modelling parameters and the frictional loss associated with the impeller in the pump can be specified on this page. The Plant+ license is required to use the Inertia features. Refer to **Section 1.5 - Plant+** for more information.

Startup Page

The **Startup** page allows the user to specify the Typical operating capacity. The Plant+ license is required to use the Startup feature. Refer to **Section 1.5 - Plant+** for more information.

6.3.4 Dynamics Tab

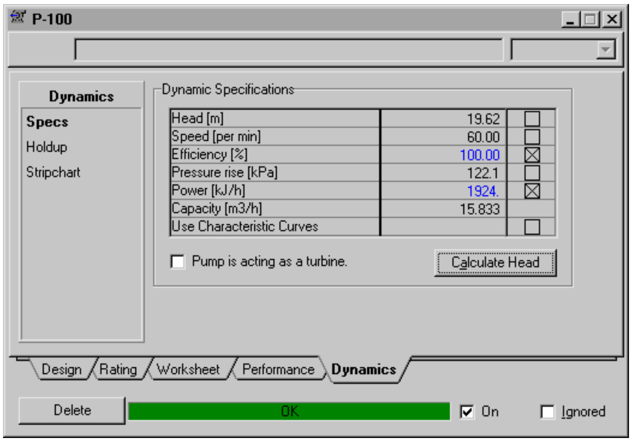
Specs Page

The dynamic specifications of the **PUMP** can be specified on the **Specs** page.

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 Increase** box should not be selected if the inlet and exit stream pressures are specified.
- The **Speed** box should not be selected if the **Use Characteristic Curves** check box is not selected.

Figure 6.23



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:

$$W = (MW)Fgh \tag{6.27}$$

where: W = the ideal pump power

MW = the molecular weight of the gas

F = the molar flow rate of the inlet stream

g = gravity acceleration

or using **Equation (6.23)**, the head is defined as:

$$h = \frac{P_2 - P_1}{\rho g} \tag{6.28}$$

If pump curves are provided in the **Curves** page of the **Ratings** tab, the ideal head can be interpolated from the flow of gas and the speed of the pump.

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:

$$\eta = \frac{W}{F(MW)(h_2 - h_1)} \tag{6.29}$$

The ideal power required by the pump is provided in **Equation (6.23)**.

Note that 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 5.3.1 - Theory** in **Section 5.3 - Pump** of the **Steady State Modeling** manual.

If pump curves are provided in the **Curves** page of the **Ratings** tab, the efficiency can be interpolated from the flow of gas and the speed of the compressor/expander.

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:

Duty = Power supplied to the fluid + Power required to change the rotational speed of the shaft + Power lost due to mechanical friction loss (6.30)

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** check box if you wish to use the curve(s) specified in the **Curves** page of the **Ratings** 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 may be specified and HYSYS will extrapolate values for head and efficiency.

Holdup Page

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

The **Stripchart** page allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the ***Create Stripchart*** button to display the stripchart.

6.3.5 Pump Example

To illustrate the use of the **PUMP** Operation, stream **FEED** will be pumped from 455 psia to 606.5 psia. The pump efficiency is 75 percent.

1. Create a fluid package using the **Peng Robinson EOS** and the following components: C1, C2, C3, i-C4, n-C4, -C5, n-C5, n-C6, n-C7, n-C8.

Property Package	Components
Peng Robinson	C1, C2, C3, i-C4, n-C4, i-C5, n-C5, n-C6, n-C7, n-C8

2. Switch to **Dynamic** mode.
3. Create a stream named **Feed** and define it as follows:

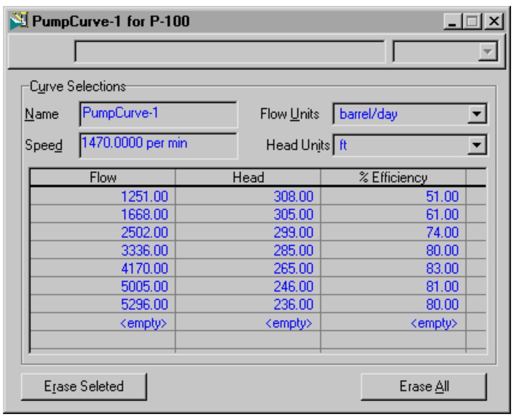
Name	FEED
Temperature [F]	250.0000
Pressure [psia]	455.0000
Molar Flow [lbmole/hr]	300.0000
Comp Mole Frac [Methane]	0.0000
Comp Mole Frac [Ethane]	0.0001
Comp Mole Frac [Propane]	0.0200
Comp Mole Frac [i-Butane]	0.1859
Comp Mole Frac [n-Butane]	0.1748
Comp Mole Frac [i-Pentane]	0.1592
Comp Mole Frac [n-Pentane]	0.1372
Comp Mole Frac [n-Hexane]	0.1613
Comp Mole Frac [n-Heptane]	0.0923
Comp Mole Frac [n-Octane]	0.0692

4. Now install the **Pump** unit operation and provide the following information:

Tab [Page]	Input Area	Entry
Design [Connections]	Inlet Outlet Energy	Feed Outlet P-100 Energy

5. Switch to the **Curves** page on the **Rating** tab.
6. Press the **Add Curve** button.
7. Complete the curve as shown in **Figure 6.24**.

Figure 6.24



8. Close the **Curve** view. Deactivate the **Efficiency** and **Pressure Rise** specifications in the **Specs** page of the **Dynamics** tab. Activate the **Use Characteristic Curves** and the **Power** specifications. Specify a pump power of 29750 Btu/hr.
9. Run the Integrator until the pump reaches steady state conditions (approximately 2 minutes, integration time.)

The results are shown in the following table.

Name	OUTLET	P-100 DUTY
Vapour Frac	0.0000	<empty>
Temperature [F]	252.3087	<empty>
Pressure [psia]	606.5	<empty>
Molar Flow [lbmole/hr]	300.000	<empty>
Mass Flow [lb/hr]	22287.0308	<empty>
Liq Vol Flow [barrel/day]	2427.8569	<empty>
Heat Flow [Btu/hr]	-2.0741e+07	29750

Note that if you clear the **On Pump Switch** check box, the conditions of stream **FEED** are directly passed to **OUTLET**.

7 Separation Operations

7.1 Vessels

All information in this section applies to every vessel operation unless others indicated.

In **Dynamics** 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 significant amount of holdup. The key differences in the vessel operations are outlined as follows:

Unit Operation	Description
Separator	The separator can have multiple feeds. There are 2 product nozzles: liquid and vapour.
3-Phase Separator	The 3-phase separator can have multiple feeds. There are 3 product nozzles: light liquid, heavy liquid, and vapour.
Tank	The tank can have multiple feeds. There is a single product nozzle which normally removes liquid 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 modelled 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.

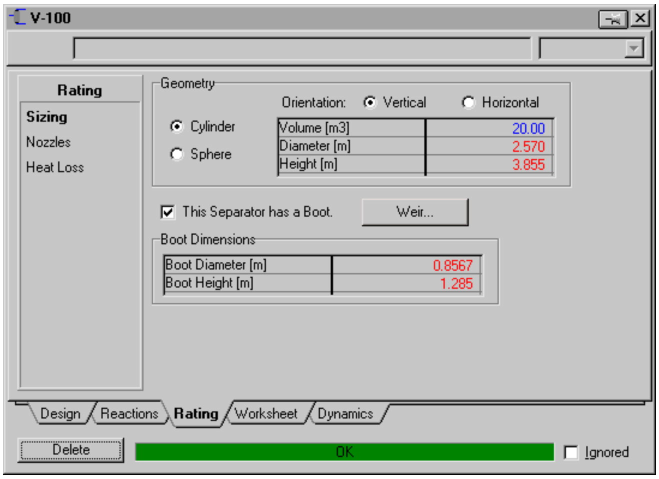
To ignore a vessel operation, select the **Ignore** check box. HYSYS will disregard the operation (and will not calculate the outlet stream) until you restore it to an active state by clearing the check box.

7.1.1 Rating Tab

Sizing Page

The **Sizing** page contains information regarding the geometry of the vessel and the boot:

Figure 7.1



Vessel Geometry

The vessel volume and shape can be specified in the **Geometry** group. The geometry of the vessel is important in determining the liquid height in the vessel. There are two possible vessel shapes as described in the table below.

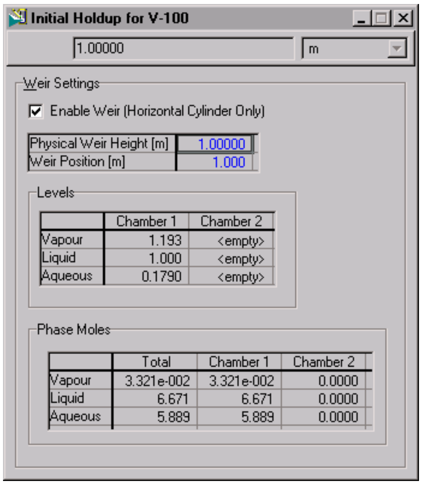
Vessel Shape	Description
Sphere	The shape of the vessel is a sphere. You can either specify the total volume or the diameter of the sphere.
Cylinder	A horizontal or vertical cylindrical vessel can be specified. You can either specify the total volume or any two of the following for the vessel: • the total volume • the diameter • the height (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 any horizontal separator by clicking the ***Weir*** button.

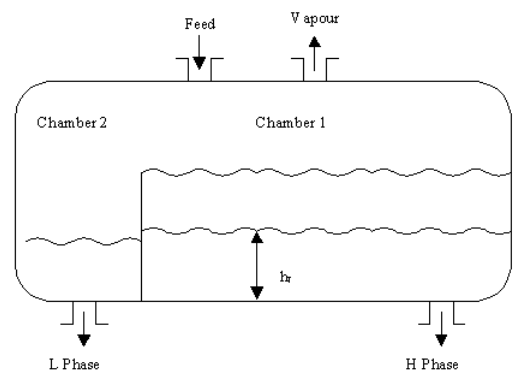
Figure 7.2



This view allows you to specify the weir Height and Position. From these inputs HYSYS will calculate the Levels and Phase Moles in each chamber.

In HYSYS the weir will simulate having two volumes inside the separator, called chamber 1 and chamber 2, but there will still only be 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 will be the same.

Figure 7.3



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** check box. The boot height is defaulted to be one third the vessel height. The boot diameter is defaulted to be one third the vessel diameter.

Nozzles Page

The **Nozzles** page contains information regarding the elevation and diameter of the nozzles. The Plant+ license is required to use the Nozzle features found on this page. Refer to **Section 1.5 - Plant+** 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 vessel 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 will draw material from the liquid holdup. If the product nozzle is located above the liquid level, the exit stream will draw 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 might 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 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 specified values:

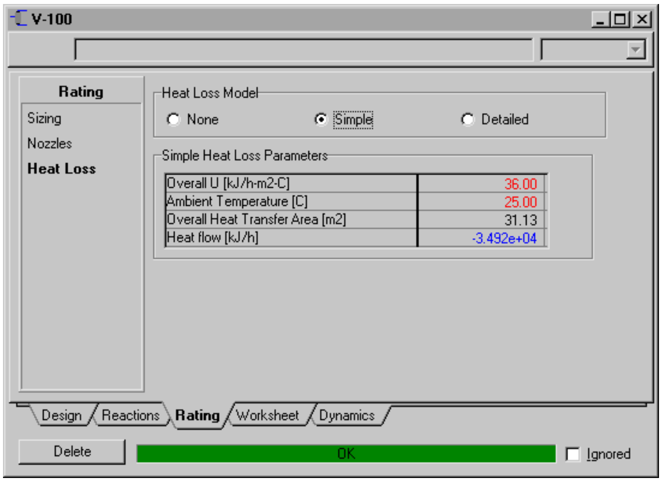
- Overall U value
- Ambient Temperature

The heat transfer area, A, and the fluid temperature, T_f, are calculated by HYSYS.Plant. The heat loss is calculated using:

$$Q = UA(T_f - T_{amb}) \tag{7.1}$$

For a separator the parameters available for Simple model are shown in **Figure 7.4**:

Figure 7.4



The simple heat loss parameters are:

- The Overall Heat Transfer Coefficient
- Ambient Temperature
- Overall Heat Transfer Area
- Heat Flow

The Heat Flow is calculated as follows:

$$Heat\ Flow = UA(T_{Amb} - T) \tag{7.2}$$

where: U = *the overall heat transfer coefficient*

A = *the heat transfer area*

T_{Amb} = *the 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, may be modified from their default values (shown in red).

Detailed Model

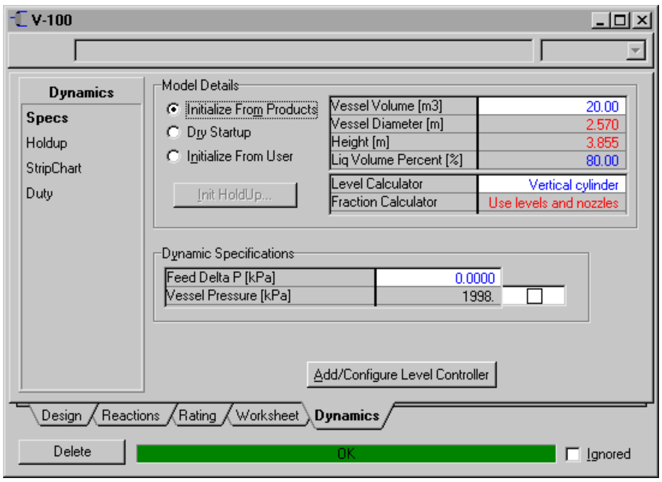
The Detailed model allows you to specify more detailed heat transfer parameters. The Plant+ license is required to use the Detailed Heat Loss model found on this page. Refer to **Section 1.5 - Plant+** for more information.

7.1.2 Dynamics Tab

Specs Page

The **Specs** page contains information regarding initialization modes, vessel geometry, and vessel dynamic specifications.

Figure 7.5



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. They are 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** cell.
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** cell 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 pressing the ***Init Holdup*** button. The liquid level is set to the value indicated in the **Liq Volume Percent** cell.

The vessel geometry can be specified in the **Model Details** section. The following vessel geometry parameters can be specified in the same manner as the **Geometry** section in **Sizing** page of the **Ratings** tab:

- Vessel Volume
- Vessel Diameter
- Vessel Height (Length)
- Vessel Geometry (Level Calculator)

Liquid Volume Percent

The liquid Volume percent is also displayed in this section. You can modify the level in the vessel at any time. HYSYS will then use that level as an initial value when the integrator is run.

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 following is a description of each Fraction Calculator option:

Fraction Calculator Mode	Description
Use Levels and Nozzles	The nozzle location and vessel liquid level affect the product composition as detailed in **Nozzles** in **Section 7.1.1 - Rating Tab**.
Emulsion Liquids	This setting is used only in tray section holdups. It should not be used anywhere else.
Trivial	This setting is used only in unit operations with negligible holdup such as a valve or pump operation. It should not be used anywhere else.

The Fraction Calculator is already correctly set for all unit operations and should not be modified by you, the user.

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** cell. The frictional pressure losses at each product nozzle are automatically set to zero by HYSYS.

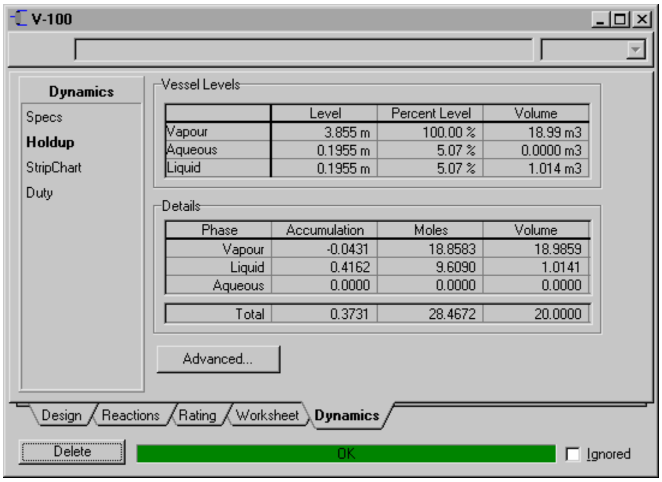
It is recommended that you enter a value of zero in the **Feed Delta P** cell because a fixed pressure drop in the vessel is not realistic for all flows. If you wish to model friction loss at the inlet and exit stream, it is suggested you add **VALVE** operations. In this case, flow into and out of the vessel will realistically be modelled.

The vessel pressure can also be specified. This specification can be made active by checking the box beside the **Vessel Pressure** cell. This specification is typically not set since the pressure of the vessel is usually variable and determined from the surrounding pieces of equipment.

Holdup Page

The **Holdup** page contains information regarding the holdup's properties, composition, and amount.

Figure 7.6



For each phase contained within the volume space of the unit operation, the following is specified:

Holdup Details	Description
Holdup Volume	The holdup volume can be specified in this cell.
Accumulation	The accumulation refers to the rate of change of material in the holdup for each phase.
Moles	These cells indicate the amount of material in the holdup for each phase.
Volume	These cells indicate the holdup volume of each phase.

Click the Advanced button to accesses a view that provides more detailed information about the holdup of that unit operation. Refer to Section 1.2.7 - Advanced Holdup Properties for more information.

Stripchart Page

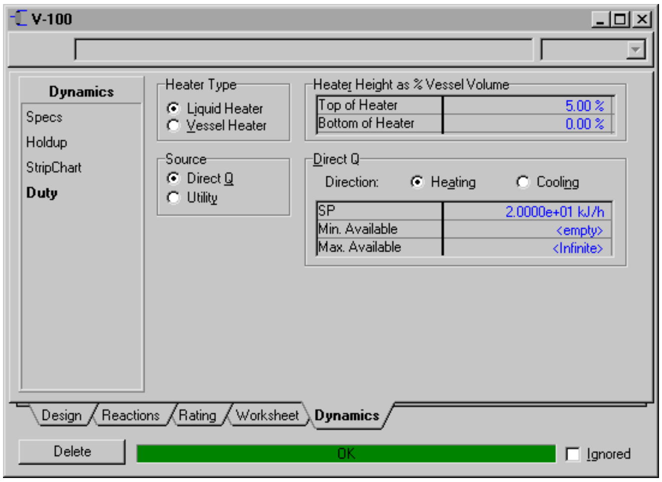
The **Stripchart** page allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the **Create Stripchart** button to display the stripchart.

Duty Page

The **Duty** page opens a list of available heating methods for the vessel operation.

The ***Duty*** *page will only be visible if a duty stream has been attached to the vessel on the* ***Connections*** *page of the* ***Design*** *tab.*

Figure 7.7



Duty Source

You can choose whether HYSYS calculates the duty applied to the vessel from a **Direct Q** option or a **Utility Fluid** option by selecting the corresponding radio button.

If the **Direct Q** source is chosen, you may directly specify the duty applied to the holdup in the **SP** cell. If the **Utility Fluid** source is chosen, you may 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** cell. For more information regarding how the utility fluid option calculates duty, see **Chapter 10 - Logical Operations**.

If the **Heating** radio button is selected, the duty shown in the **SP** field will be added to the holdup. If the **Cooling** radio button is selected the duty will be subtracted from the holdup.

Heater Type

There are two heating methods available to the general vessel operation:

If a **Vessel Heater** is chosen, 100% of the duty specified or calculated in the **SP** cell is applied to the vessel's holdup. That is:

$$Q = Q_{Total} \tag{7.3}$$

where: Q = *the total heat applied to the holdup*

Q_{Total} = *the duty calculated from the Duty Source*

If a **Liquid Heater** is chosen, the duty applied to the vessel depends on the liquid level in the tank. The heater height must be specified if the Liquid heater is chosen. 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.

$$\begin{aligned} Q &= 0 && (L < B) \\ Q &= \frac{L-B}{T-B} Q_{Total} && (B \le L \le T) \\ Q &= Q_{Total} && (L > T) \end{aligned} \tag{7.4}$$

where: L = *Liquid Percent Level (%)*

T = *Top of Heater (%)*

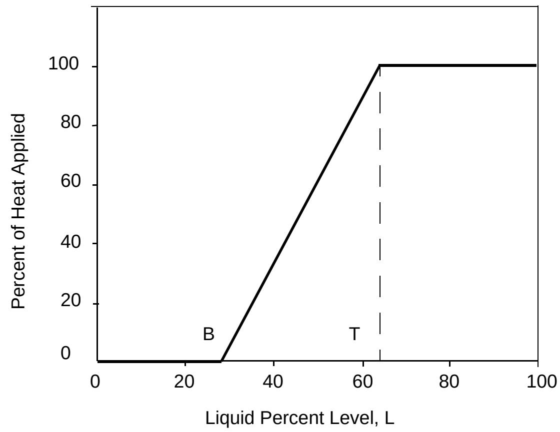
B = *Bottom of Heater (%)*

The Percent Heat Applied may be calculated as follows:

$$Percent\ Heat\ Applied = \frac{Q}{Q_{Total}} \times 100\% \tag{7.5}$$

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 7.8



Percent Heat Applied for a Liquid Heater

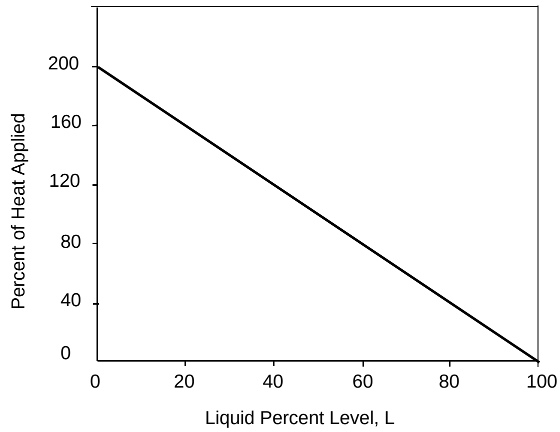
A Liquid Heater, however, is not suitable in the **CONDENSER** operation. This is because the heat transfer to the condenser depends more on the surface area of the vapour contacting the cooling coils than the liquid. Therefore, in the **CONDENSER** operation, the heat applied to the vessel holdup is expressed as follows:

$$Q = (2 - 0.02L)Q_{Total} \tag{7.6}$$

where: L= Liquid Percent Level

The heat applied to the **CONDENSER** operation directly varies with the surface area of vapour contacting the vessel wall.

Figure 7.9



Percent Heat Applied to Condenser

The following is a summary of the heating parameters for each vessel operation:

Vessel Operation	Description
Reboiler	The reboiler can only use a liquid heater.
Condenser	The condenser uses its own method of calculating the heat applied to the holdup.
Separators/ Tank/ Reactors	These unit operations can use a vessel heater or a liquid heater.

7.1.3 Separator Example

A simple **SEPARATOR** unit operation will be used to demonstrate the significance of exit stream nozzle locations on product composition and level control. The **SEPARATOR** operation will first be created in **Steady State** mode. The Dynamic Assistant will be used to size the vessel.

1. Create the following Fluid Package:

Property Package	Components
Peng Robinson	iC4, nC4, iC5, nC5, C6, C7, C8, C9

2. Create a **Feed** stream with the following properties:

Stream Name	Feed
Temperature [C]	180.00
Pressure [kPa]	2300.00
Molar Flow [kgmole/hr]	1200.00
Comp Mole Frac [i-Butane]	0.04
Comp Mole Frac [n-Butane]	0.2
Comp Mole Frac [i-Pentane]	0.1
Comp Mole Frac [n-Pentane]	0.3
Comp Mole Frac [n-Hexane]	0.25
Comp Mole Frac [n-Heptane]	0.04
Comp Mole Frac [n-Octane]	0.03
Comp Mole Frac [n-Nonane]	0.04

3. Install a **SEPARATOR** operation with the following connections:

Tab [Page]	Input Area	Entry
Design [Connections]	Name	Separator
	Feed	Feed
	Vapour Outlet	Ovhd
	Liquid Outlet	Bttms

4. Install a **VALVE** operation with the following connections:

Tab [Page]	Input Area	Entry
Design [Connections]	Name	Valve
	Feed	Bttms
	Product	Product
Design [Parameters]	Delta P	20 kPa

5. Switch to **Dynamics** mode.
6. Press the ***Dynamics Assistant*** button.

Dynamics Mode Button

Dynamics Assistant Button

A list of modifications to the flowsheet will be displayed in order to run the simulation case in **Dynamics** mode. Assume that this separator is exposed to a constant pressure source such as the atmosphere. Therefore, a pressure specification should be made on the **Ovhd** stream. The **Dynamics Assistant** will suggest that a valve be inserted on the **Ovhd** stream. However, you can instruct the **Assistant** to not make that change.

7. Select the **Insert Valves** page on the **Streams** tab. Deselect the **OK** box for the **Ovhd** stream. Click the ***Make Changes*** button in order to implement the other suggested modifications. Close the **Dynamics Assistant** view.
8. A pressure specification is required on stream **Ovhd**. Enter the **Ovhd** stream property view by double-clicking on the stream in the simulation PFD. In the **Dynamics** tab, activate the pressure specification by selecting the **Active** check box.
9. Add a **PID Controller** operation which will serve as the **SEPARATOR** level controller. Specify the following details:

Tab [Page]	Input Area	Entry
Connections	Name	SEP LC
	Process Variable Source	Separator, Liquid Percent Level
	Output Target Object	Valve
Parameters [Configuration]	Action	Direct
	Kp	1
	Ti	5 minutes
	PV Minimum	0%
	PV Maximum	100%

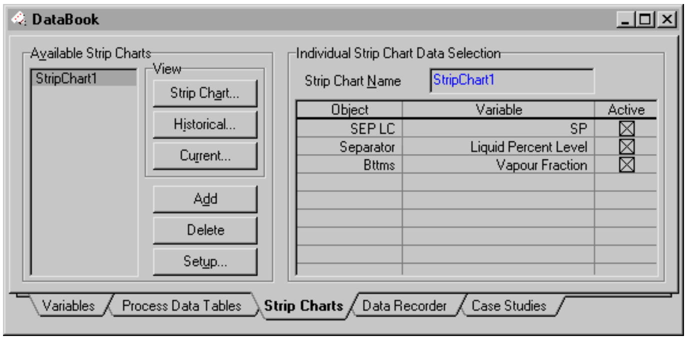
10. Click the **Face Plate** button. Change the controller mode to **Auto** on the face plate. Input a setpoint (**SP**) of 10% on the **Configuration** page of the **Parameters** tab. Run the Integrator until the separator level (**PV**) reaches its setpoint.

To examine the effect of nozzle location on product composition, add a Strip chart containing the Vapour Fraction of the **Bttms** stream and the Liquid Percent Level of the **Separator**.

11. Access the **Databook** by pressing CTRL D. Press the ***Insert*** button on the **Variables** tab to open the Variable Navigator. From the **Flowsheet** group, select the **Liquid Percent Level** of the **Separator** operation. Press the ***Insert*** button again and select the **Vapour Fraction** of stream **Bttms**. Insert the setpoint (**SP**) of the **SEP LC** Controller.

12. In the **Strip Charts** tab, select the ***Add*** button in the **Available Strip Charts** group. Activate the **Active** check box for the three listed variables.

Figure 7.10



13. Press the ***Strip Chart*** button to access the **Strip Chart** view.

14. In order to modify the displayed range of the Strip chart variables, double click on any area within the Strip Chart view. In the **Numerical Line Properties** tab of the **Strip Chart Setup** property view, modify the strip chart as follows:

Tab [Page]	Object [Variable]	Line Minimum	Line Maximum
Numerical Line Properties	SEP LC [Setpoint]	0	50
	Separator [Liquid Percent Level]	0	50
	Bttms [Vapour Fraction]	0	1

15. Run the Integrator for approximately 10 integration minutes or until all the variables line out. This can easily be observed in the strip chart.

16. Now, modify the **Bttms** exit stream location from the **Separator** vessel. Enter the **Separator** property view. The exit stream nozzle locations are displayed on the **Nozzles** page of the **Rating** tab. In the Nozzle Parameters group box, modify the **Elevation (% of Height)** field of stream **Bttms** from 0% to 25%.

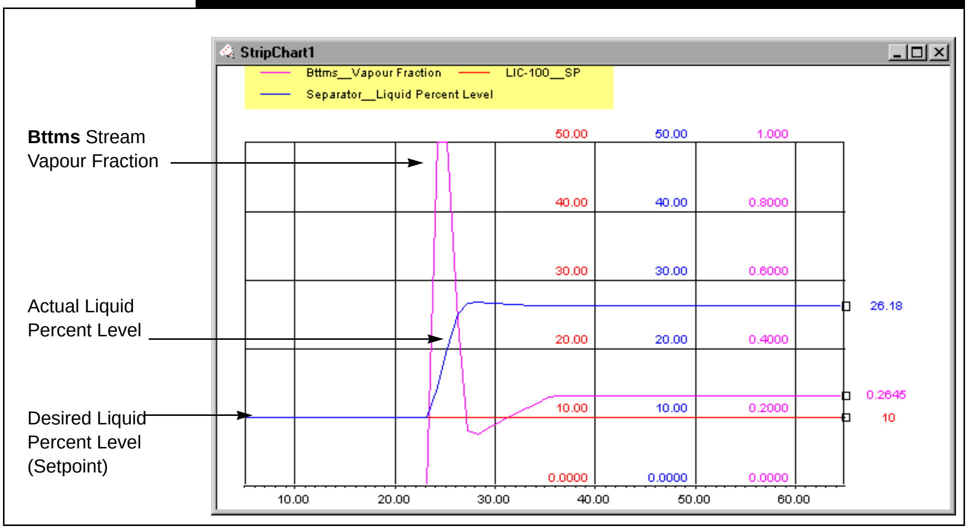
Figure 7.11

Nozzle Parameters

	Feed	Ovhd	Bttms
Diameter [m]	0.2519	0.2519	0.2519
Elevation (Base) [m]	2.519	5.038	1.259
Elevation (Ground) [m]	2.519	5.038	1.259
Elevation (% of Height) [%]	50.00	100.00	25.00

17. Run the Integrator. The level percent level of the **Separator** cannot be maintained at the 10% setpoint demanded by the **LC SEP** Controller. This is because the nozzle location is above the desired liquid level of the **Separator**. The stripchart is as displayed as follows:

Figure 7.12



The vapour fraction of stream **Bttms** increases immediately to one after the integrator is run. Since the initial liquid level is below the **Bttms** nozzle location, only vapour is drawn. Because the nozzle location is located above the setpoint, the **SEP LC** Controller cannot maintain a liquid level of 10%. The **Valve** operation is driven to 100% opening and the liquid level in the **Separator** comes to equilibrium at 26%.

18. The **Bttms** and **Ovhd** stream conditions are displayed in the **Conditions** page of the **Worksheet** tab.

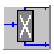
Figure 7.13

Name	Feed	Bttms	Ovhd
Vapour	0.3117	0.2645	1.0000
Temperature [C]	180.0000	179.9997	179.9997
Pressure [kPa]	2300.	2300.	2300.
Molar Flow [kgmole/h]	1200.0000	1123.0155	76.9846
Mass Flow [kg/h]	92304.1874	86715.7805	5588.4018
LiqVol Flow [m3/h]	144.4126	135.5115	8.9011
Molar Enthalpy [kJ/kgmole]	-1.455e+05	-1.465e+05	-1.307e+05
Molar Entropy [kJ/kgmole-C	180.5	175.6	191.4
Heat Flow [kJ/h]	-1.7459e+08	-1.6453e+08	-1.0058e+07

7.2 Component Splitter

For information on setting up a Component Splitter refer to Chapter 6 in the Steady State Modelling Guide.

With a **COMPONENT SPLITTER**, a material feed stream is separated into two component streams based on the parameters and split fractions that you supply. You are required to 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.



Component Splitter Button

To install the **COMPONENT SPLITTER** operation, press **F12** and select **Component Splitter** from the **UnitOps** view or select the ***Component Splitter*** button from the Object Palette.

7.2.1 Rating Tab

Nozzles Page

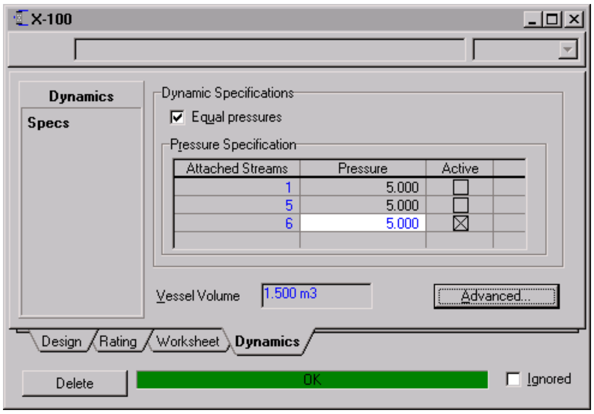
The **Nozzles** page contains information regarding the elevation and diameter of the nozzles. The Plant+ license is required to use the Nozzle features found on this page. Refer to **Section 1.5 - Plant+** for more information.

7.2.2 Dynamics Tab

Specs Page

The **Specs** page contains information regarding pressure specifications of the streams.

Figure 7.14



The **Equal Pressures** check box 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.

8 Column Operation

*For more information regarding the Column Template, Refer to the **Steady State Modelling Guide, Chapter 7 - Column**.*

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 a 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.

8.1 Theory

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:

$$L_N = C\rho l_w h^{1.5} \tag{8.1}$$

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:

$$F_{vap} = k\sqrt{\Delta P_{friction}} \tag{8.2}$$

where: Fvap= Vapour flowrate leaving tray N

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

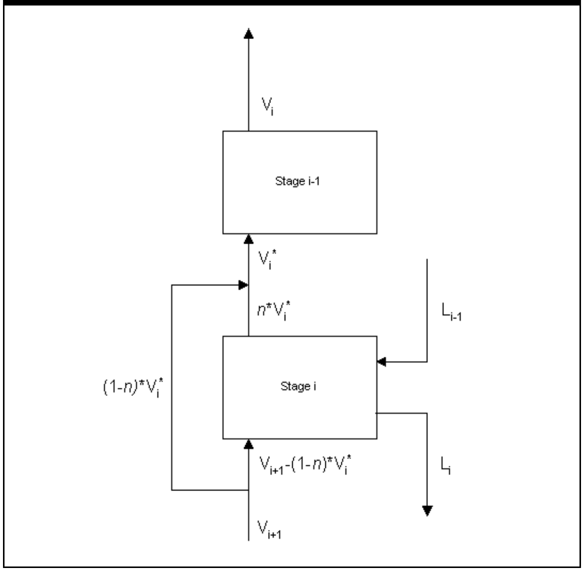
$\Delta P_{friction}$ *= Dry hole pressure drop*

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 **Figure 8.1**, where *n* is the specified efficiency.

Figure 8.1



HYSYS.Plant has the ability to model both weeping and flooding inside the column. If $\Delta P_{friction}$ is very small, the stage will exhibit 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 will approach 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.

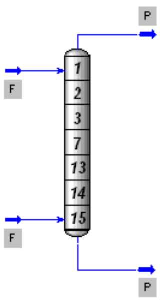
8.2 Pressure Flow

In the following sections, the pressure flow specifications presented are the recommended configurations if no other equipment is connected. Other combinations of pressure flow specifications are possible, however they may lead to less stable configurations.

Regardless of the pressure flow specification configuration, when performing detailed dynamic modelling 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.

8.2.1 Columns

Absorber

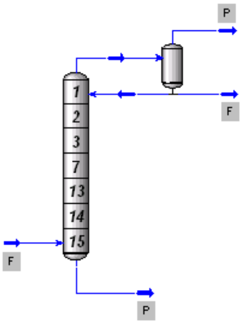


The basic **Absorber** column has two inlet and two exit streams. When used alone, the **Absorber** has four boundary streams and so requires four Pressure Flow specifications. A pressure specification will always be required for the liquid product stream leaving the bottom of the column. A second pressure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

The column to the left 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 will need 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

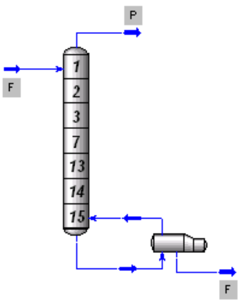


The basic **Refluxed Absorber** column has a single inlet and two or three exit streams, depending on the condenser configuration. When used alone, the **Refluxed Absorber** has three or four boundary streams (depending on the condenser) and requires four or five pressure-flow specifications; generally two pressure and three flow specifications. A pressure specification will always be 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 8.2.2 - Condensers and Reboiler**.

The column to the left, 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 will need 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.

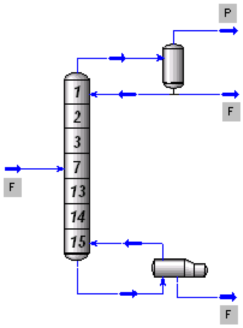
Reboiled Absorber



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 will always be required for the vapour product leaving the column.

The column to the left 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 8.2.2 - Condensers and Reboiler.

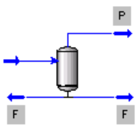
The column to the left shows the recommended pressure-flow specifications for a stand alone distillation column with a partial condenser.

8.2.2 Condensers and Reboiler

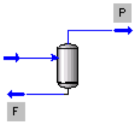
The following sections provide some recommended pressure-flow specifications for simple dynamic modelling only. The use of flow specifications on reflux streams is not recommended for detailed modelling. If the condenser liquid level goes to zero, a mass flow specification will result in a large volumetric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

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

Partial Condenser

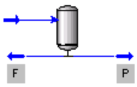


The partial condenser has three exit streams; the overhead vapour stream, the reflux stream, and the distillate stream. All three exit streams must be specified when attached to the main tray section. One pressure specification is recommended for the vapour stream, and one flow specification for either of the liquid product streams. The final pressure flow specification may 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 may be specified.



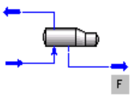
Fully-Refluxed Condenser

The Fully-Refluxed condenser has two exit streams: the overhead vapour stream and the reflux stream. A pressure specification is required for the overhead vapour stream and a flow specification is required for the reflux stream.



Fully-Condensed Condenser

A Fully-Condensed condenser has two exit streams: the reflux stream and the distillate stream. 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 stream: the boilup vapour stream and the bottoms liquid stream. 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.

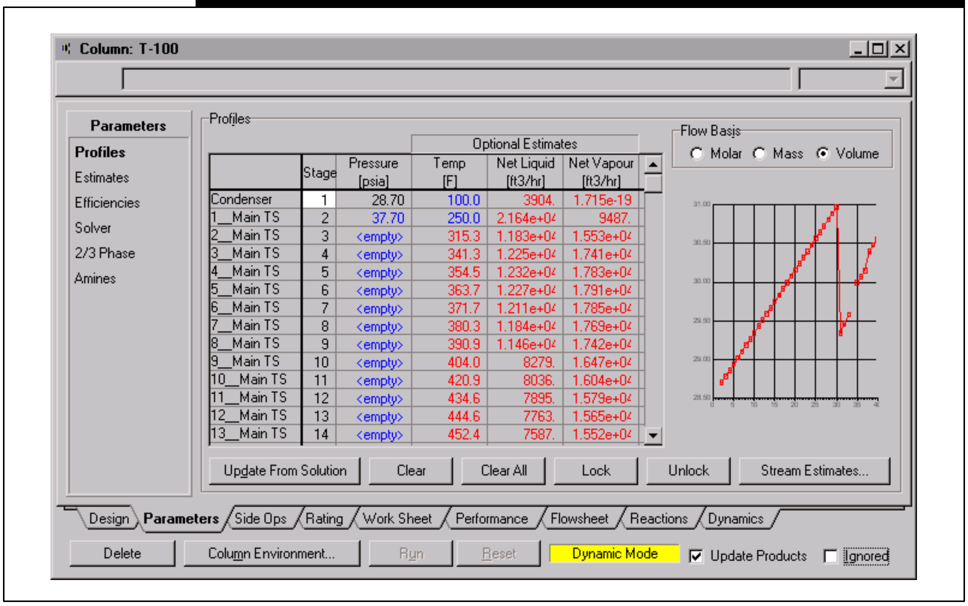
8.3 Column Runner

The Column Runner property view contains a summary of the most important details of the column that are required for a stable dynamic model. More detailed information in available in the Tray Section property view (discussed in **Section 8.4 - Tray Section**) and the Condenser and Reboiler property views (discussed in **Chapter 7 - Separation Operations**).

8.3.1 Parameters Tab

Profiles Page

Figure 8.2

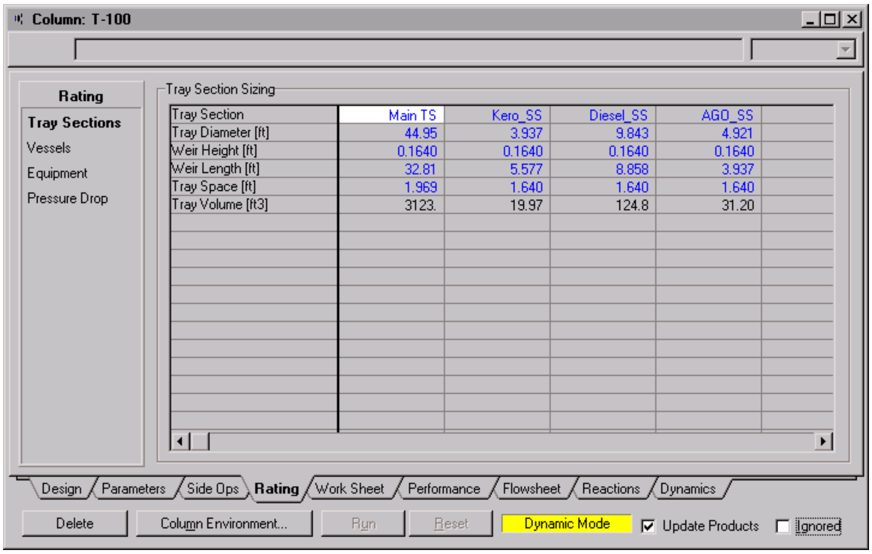


Although the **Profiles** page is mainly used for steady state simulation, it does contain vital information for running a column in dynamics. One of the most important aspects of running a column in dynamics is the pressure profile. While a steady state column can run with zero pressure drop across 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 dynamics, mode, the pressure profile can be obtained from the streams if not directly supplied. In either case, the closer the initial pressure profile is to the one calculated while running in dynamics, the fewer problems you will encounter.

8.3.2 Rating Tab

Tray Sections Page

Figure 8.3



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

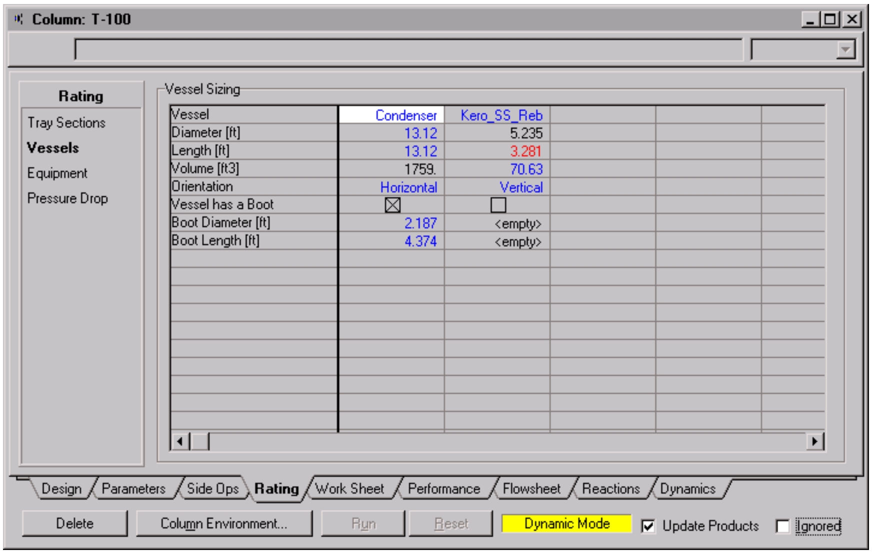
The **Tray Sections** page contains all the required information for correctly sizing the column tray sections. The tray section diameter, weir length, weir height, and the tray spacing are required for an accurate and stable dynamic simulation. You must supply all the information on this page. With the exception of the Tray Volume, no other calculations are performed on this page.

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 8.4



Equipment Page

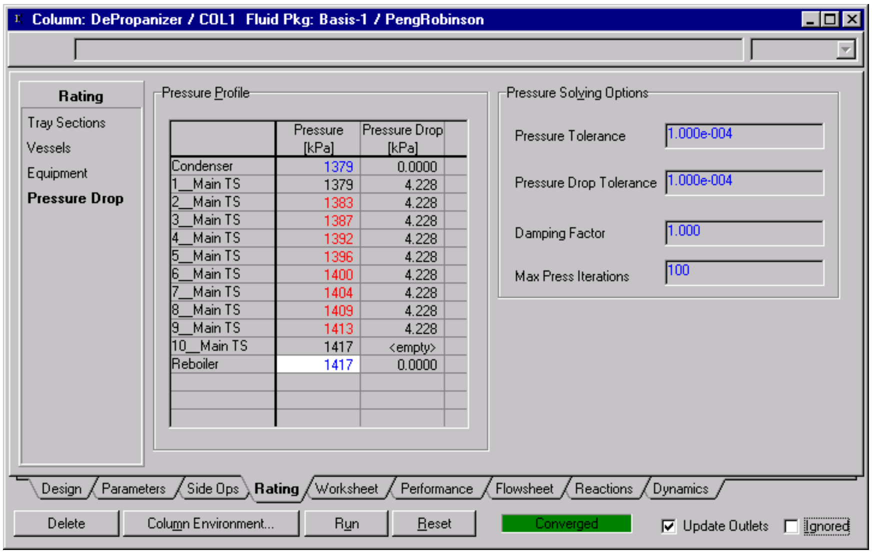
The **Equipment** page contains a list of all the additional equipment which is part of the column subflowsheet. The list does not contain equipment which is part of the original template. Any extra equipment which is added to the subflowsheet (pump arounds, side strippers, etc.) is listed here. Double clicking on the equipment name will open it's property view on the **Ratings** tab.

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 may 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

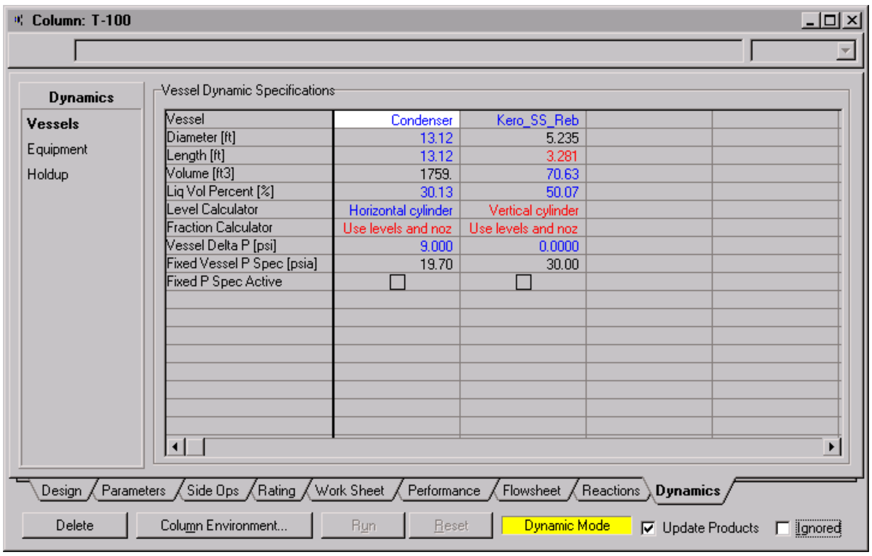
Figure 8.5



8.3.3 Dynamics Tab

Vessels Page

Figure 8.6



The **Vessels** page contains a summary of the sizing information for the different vessels contained in the Column Sub-Flowsheet. In addition, it contains the possible dynamic specifications for these vessels.

Equipment Page

The **Equipment** page on the **Dynamics** tab displays the same information as the **Equipment** page on the **Ratings** tab. The difference is that double clicking on the equipment name will open its property view on the **Dynamics** tab.

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.

8.4 Tray Section

8.4.1 Rating Tab

Sizing Page

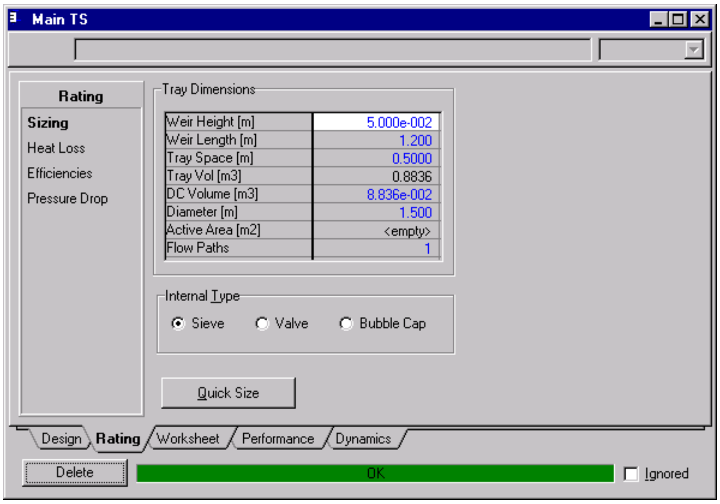


Figure 8.7

The **Sizing** page contains all the required information for correctly sizing the column tray sections. The tray section diameter, weir length, weir height, and the tray spacing are required for an accurate and stable dynamic simulation. You must supply all the information on this page, HYSYS will only calculate the tray volume, based on the weir length, tray spacing and tray diameter.

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

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

When simulating swedged columns, the column must be simulated with multiple tray sections. A tray section can only have one diameter, weir length and weir height. Therefore when simulating a swedged column, a new tray section will need to be installed for each column diameter.

Nozzles Page

The **Nozzles** page contains the elevations at which vapour and liquid enter or leave the tray section. The Plant+ license is required to use the Nozzles features found on this page. Refer to **Section 1.5 - Plant+** for more information.

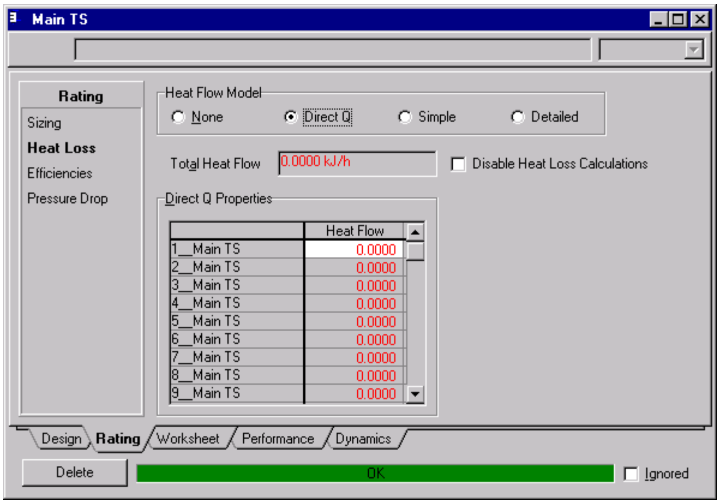
Heat Loss Page

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 tray sections.

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 each individual tray section.

Figure 8.8

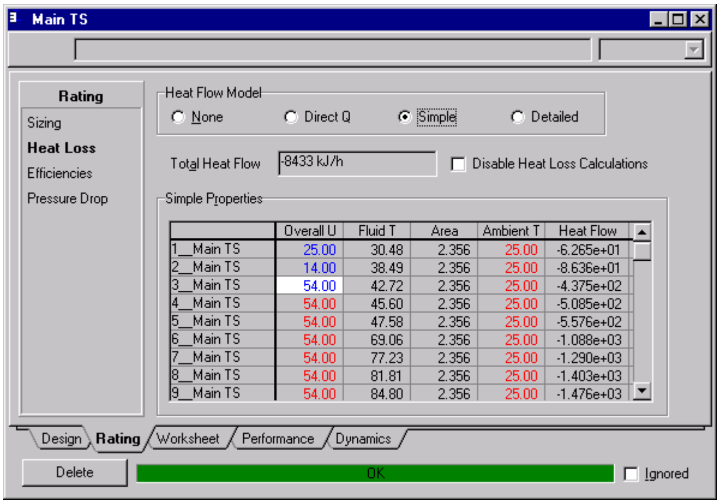


Simple Heat Loss Model

The Simple model allows you to calculate the heat loss from these specified values:

- Overall U value
- Ambient Temperature

Figure 8.9



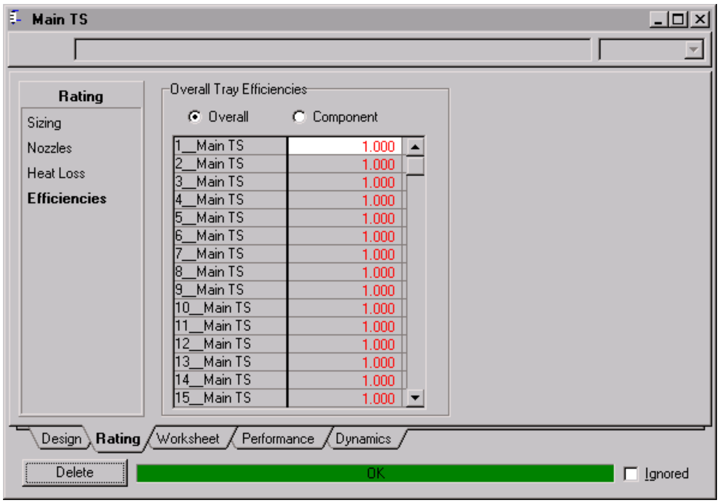
Detailed Heat Loss Model

The Detailed model allows you to specify more detailed heat transfer parameters. The Plant+ license is required to use the Detailed Heat Loss model. Refer to **Section 1.5 - Plant+** for more information.

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.

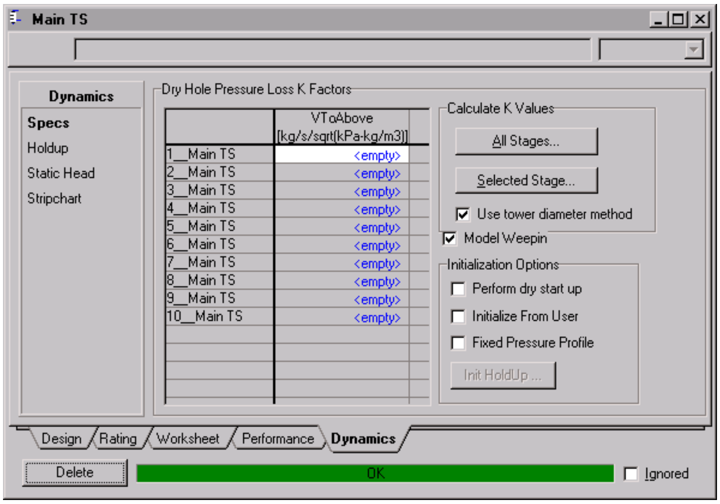
Figure 8.10



8.4.2 Dynamics Tab

Specs Page

Figure 8.11



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 pressing the ***All Stages*** button. If you want HYSYS to calculate the k values for certain trays only, highlight the desired trays and press the ***Selected Stages*** button. HYSYS will only calculate the k values for the selected stages.

The **Use tower diameter method** check box, when checked, calculates the k values for the column based on the column diameter. When the radio button is unchecked 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** check boxed, when checked, will take into account any weeping that occurs on the tray sections and add the effects to your model.

The **Perform dry start up** check box allows you to simulate a dry start up. Checking this box will remove all the liquid from all the trays when the integrator is started.

The **Initialize From User** check box allows you to start the simulation from conditions you specify. Checking this box will activate 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** check box allows you to simulate the column based on the fixed pressure profile.

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 view. Double clicking on a a stage name in the Holdup column will open the stage property view, which is discussed in the following section.

You can double click on any cell within each row to view the advanced holdup properties for each specific tray section.

Static Head Page

The Plant+ license is required to use the Static Head features. Refer to **Section 1.5 - Plant+** for more information.

Stripchart Page

The **Stripchart** page allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the **Create Stripchart** button to display the stripchart.

8.5 Column - Pressure Profile Example

A new feature has been added to the Dynamic column. It will simplify the column solution for some novice users who were experiencing problems with the tray section pressure. Now you can have a constant pressure profile options for tray sections in dynamics. It means using the same steady state pressure for dynamics simulation as well.

The pressure flow equations are still used to solve the column, but the holdup pressure on each stage is specified rather than calculated. This is a user option and can be turned off by default. The fixed pressure profile mode will apply to stages in a tray section.

8.5.1 Quick Start

To explain the new steady state pressure profile feature in dynamic, a 20 stage CO2 Rejection Tower will be installed.

Starting the Simulation

1. Create a new case and add a fluid package. Select NRTL as the property package and include the components listed in the following table.

Property Package	Components
NRTL	Ethanol, H2O, CO2.

2. On the **Binary Coeffs** page of the Fluid Package use UNIFAC VLE and click the ***Unknowns Only*** button to estimate the missing interaction parameters.
3. Access the **Session Preferences** view by selecting **Preferences** on the **Tools** menu. On the Units page of the variable tab, change the unit set to **Field** units. Enter the Simulation Environment and create the streams **Wash_H2O** and **To_CO2Wash**, they will be the feed to the column:

Stream Name	Wash_H2O	To_CO2Wash
Temperature [F]	77	86
Pressure [psia]	14.7	16.8
Molar Flow [lbmole/hr]	286.6	146.5
Comp Mole Frac [Ethanol]	0.0000	0.0170
Comp Mole Frac [H2O]	1.0000	0.0409
Comp Mole Frac [CO2]	0.0000	0.9421

4. Before installing the column, select **Preferences** from the HYSYS **Tools** menu. On the **Options** page of the **Simulation** tab, ensure that the **Use Input Experts** checkbox is selected (checked), then ***Close*** the view.
5. Install an absorber named **CO2Wash** using the following information:

ABSORBER [CO2WASH]		
Tab [Page]	**Input Area**	**Entry**
Connections	No. of Stages	20
	Feed Streams (Stage)	Wash_H2O (Top Stage) To_CO2Wash (Bottom Stage)
	Ovhd Vapour	CO2_Stream
	Bottoms Liquid	Liquid_Bttms
Pressure Profile	Stage 1	14.7 psia
	Stage 20	16.8 psia

6. Press the ***Run*** button on the Column property view to converge the column.
7. Enter the column environment by clicking on the **Column Environment** button.
8. Install a Separator named **Sump** using the following information:

SEPARATOR [SUMP]		
Tab [Page]	**Input Area**	**Entry**
Design [Connections]	Vapour Outlet	Vapour_Out
	Liquid Outlet	Liquid_Out
Design [Parameters]	Delta P	0.0000 psi

Using the PFD tools perform the following steps.

*Refer to **Chapter 3 - PFD** for information on how to use the PFD tools.*

9. Break the connection between the **To_CO2Wash** stream and the absorber and attach the stream to the separator inlet.
10. Attach the **Liquid_Bttms** stream from the absorber to the separator inlet.
11. Attach the **Vapour_Out** stream from the top of the separator to the bottom stage of the absorber.

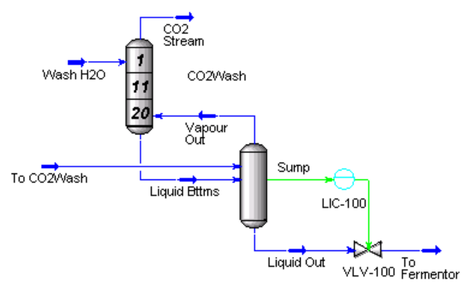
12. Install a Valve using the following information:

VALVE [VLV-100]		
Tab [Page]	**Input Area**	**Entry**
Design [Connections]	Inlet	Liquid_Out
	Outlet	To_Fermentor
Design [Parameters]	Delta P	0.0000 psi

13. Install a PID Controller using the following information:

PID CONTROLLER [LIC-100]		
Tab [Page]	**Input Area**	**Entry**
Connections	Process Variable Source	Sump - Liquid Percent Level
	Output Target Object	VLV-100
Parameters [Configuration]	Action	Direct
	Mode	Auto
	Ti	10
	Pv Minimum	0.0000 %
	Pv Maximum	100.0000 %

Figure 8.12



14. Return to the Parent Simulation Environment and **run** the simulation.

Moving to Dynamics

The **Dynamics Assistant** suggests changes to the steady state case such that it can run in dynamics environment. It is recommended to review these changes and accept some or all of them.

15. Run the **Dynamics Assistant** to enable all the specifications required to run in dynamics mode. On the **Dynamics Assistant** view, accept the default changes by pressing the ***Make Changes*** button.
16. Switch to the dynamics mode by pressing the ***Dynamics Mode*** button. Press the ***Yes*** button on the confirmation view.
17. In the column environment, open the tray section view and move to the **Specs** page on **Dynamics** tab. Select the **Fixed Pressure Profile** check box under the **Initialization Options** group.

Integrator *buttons*

18. Run the Integrator by pressing the ***Integrator*** button. At this stage the **Dynamic Assistant** will detect that the **Fixed Pressure Profile** option has been selected and it will ask for confirmation. Press the ***No*** button to run the integrator with the selected option. Observe the pressure profile on the **Pressure** page of the **Design** tab on the tray section view. Notice that the tray section pressure will not change.

8.5.2 Pressure Profile

The **Fixed Pressure Profile** check box allows you to run the column in dynamics mode using the stead state pressure profile. This option simplifies the column solution for inexperienced users and makes their transition from the stead state to dynamics simulation a bit easier.

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 (8.3)**.

$$\text{flow} = k \times \sqrt{\text{density} \times \text{friction pressure losses}} \qquad (8.3)$$

In HYSYS, the k-value is calculated by assuming:

$$k \alpha (\text{Tray diameter})^2 \qquad (8.4)$$

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 may be known. This allows the k-values to be directly calculated to match steady state results more closely.

The **Adjusted for steady state** option under the **Calculate K Values** group can be checked on the **Specs** page of the **Dynamics** tab, on the **Tray Section** view before hitting either ***All Stages*** or ***Selected Stages*** button. It calculates the new k-values right before the tray section is run in dynamics for the first time. This option is most useful when the case has just been converted from steady state and the user is satisfied with all the values. It only considers steady state results and may therefore not be fully applicable if the case and operating conditions have changed substantially since steady state or there was no steady state solution.

8.6 A Column Tutorial

The DeButanizer column setup in this tutorial presents a general procedure for simulating towers and can be adapted to model other types and configurations of columns in *HYSYS.Plant*. A more detailed dynamic simulation of this column is presented in **Dynamic Debutanizer Tutorial** in the **Tutorial Book**.

Figure 8.13

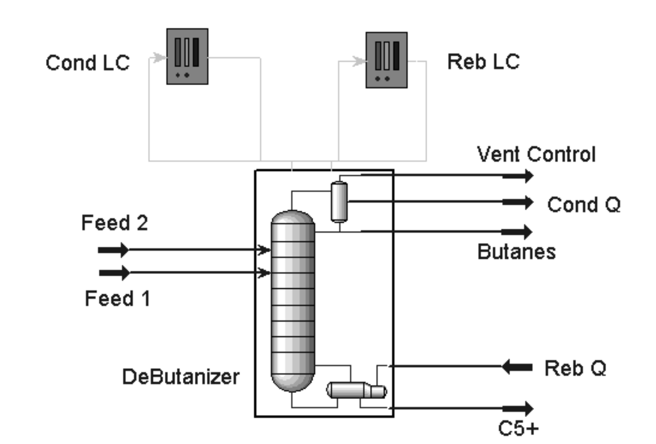
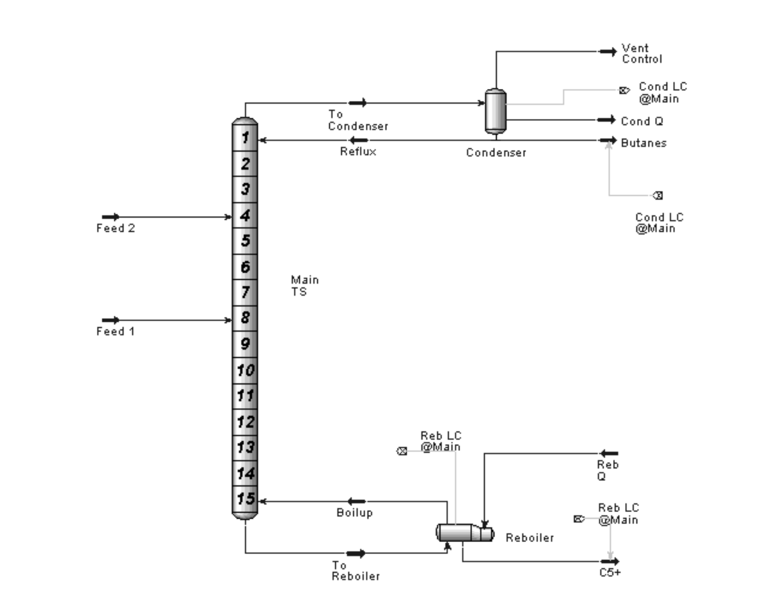


Figure 8.14



8.6.1 Beginning the Simulation

1. Start a new case.
2. Create a fluid package using the following property package and components:

Property Package	Components
Peng Robinson	Propane, i-Butane, n-Butane, isobutene, i-Pentane, n-Pentane, n-Hexane, n-Heptane, n-Octane

3. Enter the Simulation Environment.

4. Create the two feed streams to the tower:

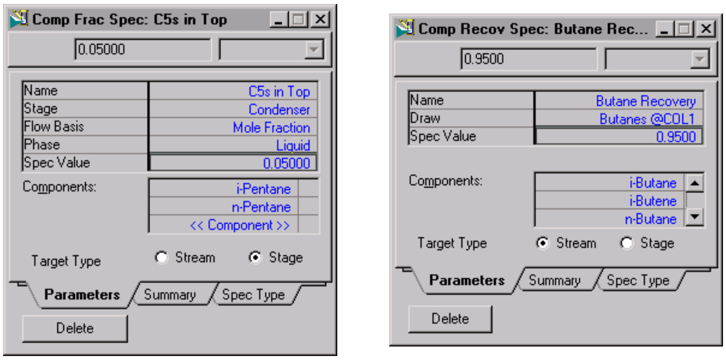
Stream Name	Feed 1	Feed 2
Temperature [F]	300.0	140.0
Pressure [psia]	220	350
Mass Flow [lb/hr]	18000	9000
Mass Fraction [propane]	0.012	0.020
Mass Fraction [i-butane]	0.170	0.190
Mass Fraction [n-butane]	0.170	0.200
Mass Fraction [i-butene]	0.008	0.220
Mass Fraction [i-pentane]	0.140	0.160
Mass Fraction [n-pentane]	0.140	0.210
Mass Fraction [n-hexane]	0.110	0.000
Mass Fraction [n-heptane]	0.130	0.000
Mass Fraction [n-octane]	0.120	0.000

5. Add a **DISTILLATION COLUMN** to the case. Provide the following information:

<table>
<tr><th>Tab [Page]</th><th>Input Area</th><th>Entry</th></tr>
<tr><td rowspan="13">Design [Connections]</td><td>Column Name</td><td>DeButanizer</td></tr>
<tr><td>Number of Stages</td><td>15</td></tr>
<tr><td>Feed Stream (Feed Stage)</td><td>Feed 2 (4)</td></tr>
<tr><td>Feed Stream (Feed Stage)</td><td>Feed 1 (8)</td></tr>
<tr><td>Condenser Type</td><td>Partial</td></tr>
<tr><td>Ovhd Vapour</td><td>Vent Control</td></tr>
<tr><td>Ovhd Liquid</td><td>Butanes</td></tr>
<tr><td>Bottoms Liquid</td><td>C5+</td></tr>
<tr><td>Reboiler Duty</td><td>Reb Q</td></tr>
<tr><td>Condenser Duty</td><td>Cond Q</td></tr>
<tr><td>Condenser Pressure [psia]</td><td>205</td></tr>
<tr><td>Condenser Delta P [psi]</td><td>3</td></tr>
<tr><td>Reboiler Pressure [psia]</td><td>215</td></tr>
<tr><td>Design [Monitor]</td><td>Ovhd Vap Rate Specified Value [lbmole/hr]</td><td>0</td></tr>
</table>

6. On the **Specs** page of the **DeButanizer** property view, add a **Component Fractions** specification and a **Component Recovery** specification. The completed views for the specifications are shown in **Figure 8.15**:

Figure 8.15



7. On the **Monitor** page of the **Debutanizer** property view, deactivate the **Active** check box for both the **Reflux Ratio** and **Distillate Rate** specifications.
8. Activate the **Active** check box for both new specifications, **C5s in Top** and **Butane Recovery**.

The column will begin solving.

9. Tighten the purities on the tower by changing the specified values of **C5s in Top** and **Butane Recovery** to 0.018 and 0.951 respectively.

The tower is producing a higher purity product, but the trade-off is increased operating costs arising from higher condenser and reboiler duties. On the **Monitor** page of the **Design** tab, note the steady state reflux ratio.

10. Save the case as *debSS.hsc*.

8.6.2 Dynamic Specifications

Dynamic Mode Button

1. Open the case *debSS.hsc*, if it is not already open.
2. Switch to dynamic mode.

To summarize the pressure-flow specifications for this case, one pressure-flow specification will be set around the condenser and one will be set on each of the flowsheet boundary streams:

A condenser for a refluxed tower increases the flowsheet degrees of freedom by one and thus, requires an additional pressure-flow specification.

- Feed 1
- Feed 2
- Vent Control
- Butanes
- C5+

In addition, the condenser, reboiler and tray section will be sized, using the steady state flows for the design.

Condenser Specification

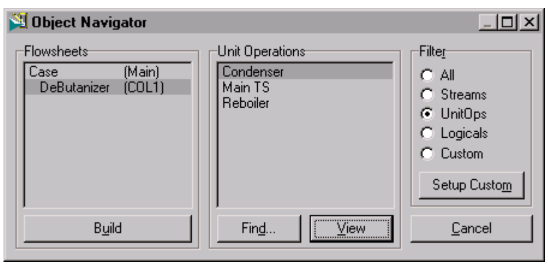
For the flowsheet created in this example, the liquid product stream from the condenser, **Butanes**, is a flowsheet boundary stream and will have a flowrate specification. With this in mind, one other condenser specification will define the pressure-flow relationship within the condenser. You have two options for this specification:

- Flow of the stream Reflux
- Reflux Rate as a ratio of total liquid from the unit

For this case, the reflux rate as a ratio of total liquid will be specified.

3. Open the **Object Navigator** by pressing **F3**.
4. Select the **UnitOps** radio button in the **Filter** group.
5. Select **DeButanizer** in the **Flowsheets** group and **Condenser** in the **Unit Operations** group.

Figure 8.16



6. Press the **View** button.

7. On the **Condenser** property view, select the **Dynamics** tab, **Specs** page.

Since the steady state reflux ratio was approximately 5.5, the rate of **Reflux** was 4.5 times the rate of **Butanes**. There was no overhead vapour so the reflux ratio compares only the two liquid streams. By dividing 4.5/5.5, the reflux rate as a ratio of total liquid flow is obtained.

8. For the **Reflux flow/total liq flow** specification, activate the corresponding check box and input **0.82**.

Sizing the Condenser

Using the flows that were calculated in steady state as a design basis, the condenser will now be sized.

9. Select the **Worksheet** tab, **Conditions** page on the **Condenser** property view and make a note of the liquid volume flow of the stream **To Condenser**.

For a quick sizing calculation, use a flow of 1750 ft3/hr, a 20 minute residence time and a liquid operating level of 50%.

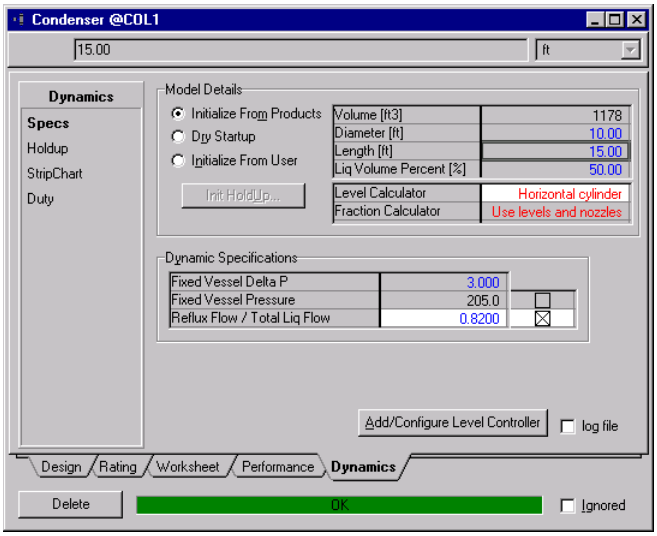
You can change the barrel/day unit by cloning the Field unit set and changing the Liquid Volume Flow unit to ft3/hr.

$$1750\frac{ft3}{hr} \times \frac{1hr}{60min} \times 20min \times \frac{1}{0.5} = 1167ft3$$

10. On the **Dynamics** tab, **Specs** page of the **Condenser** property view, delete the default **Volume**.
11. Input a **Diameter** of 10 ft and a **Length** of 15 ft to obtain a vessel volume that approaches 1178 ft3.

The completed **Specs** page should appear as shown:

Figure 8.17



Condenser Utility Valve

12. Select the **Duty** page on the **Dynamics** tab of the **Condenser** property view.
13. Press the **Initialize Duty Valve** button.

This initialization provides a UA value, a flow range for the utility fluid and an inlet temperature for the fluid according to the current flow of material into the condenser.

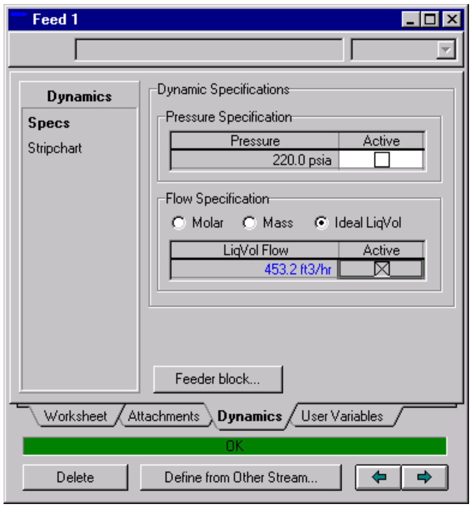
Boundary Stream Specifications

For the stream **Feed 1**, set a dynamic flow specification on a volume basis as follows:

14. Open the property view for **Feed 1**.
15. Select the **Dynamics** tab.
16. Deactivate the **Pressure** specification.
17. Change the **Flow** specification basis to **Ideal LiqVol**, by selecting the corresponding radio button.

The **Dynamics** tab should appear as in Figure 8.18:

Figure 8.18



18. Repeat steps #15 to #17 for **Feed 2**.

Now you can set a volume basis dynamic flow specification on stream **Butanes**. Proceed through the following steps:

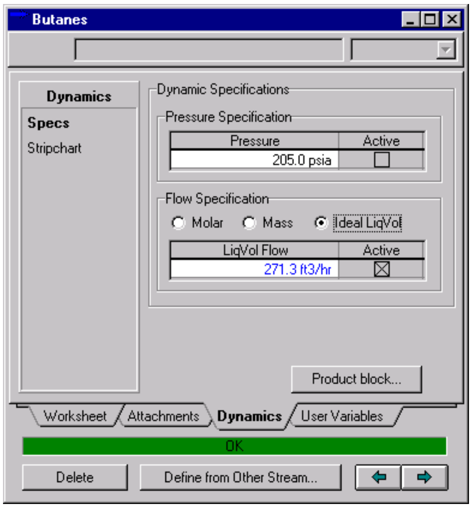
19. Open the property view for stream **Butanes**.
20. Choose the **Dynamics** tab.

Since the pressure and flow of stream **Butanes** were not specified in steady state, neither is used as an active pressure flow specification.

21. Activate the **Flow** specification and select **Ideal LiqVol** as the flow basis.

The **Dynamics** tab should appear as shown in **Figure 8.19**:

Figure 8.19



22. Repeat steps #20 and #21 for the stream **C5+**.

The last pressure-flow specification will be a dynamic pressure specification on the stream **Vent Control**.

23. Open the property view for the stream **Vent Control**.
24. Select the **Dynamics** tab.
25. Activate the **Pressure** specification.

If you open the **Workbook** to the **Material Streams** tab, you will notice that the only pressure and flow values that are shown in blue are the ones that were just selected as dynamic specifications.

Reboiler Specifications

Using the flows that were calculated in steady state as a design basis, the reboiler, will now be sized.

26. Press F3 to open the **Object Navigator**.
27. In the **Flowsheets** group, select **DeButanizer**.
28. In the **Unit Operations** group, choose **Reboiler**.
29. Press the **View** button.

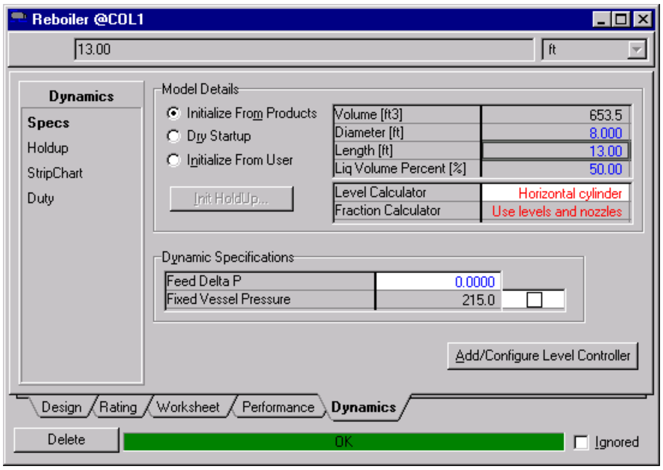
30. Select the **Worksheet** tab, **Conditions** page on the **Reboiler** property view and make a note of the liquid volume flow of the stream **To Reboiler**.

For a quick sizing calculation, use a flow of 2000 ft3/hr, a 10 minute residence time and a liquid operating level of 50%.

$$2000\frac{ft3}{hr} \times \frac{1hr}{60min} \times 10min \times \frac{1}{0.5} = 667ft3$$

31. On the **Dynamics** tab, **Specs** page of the **Reboiler** property view, delete the default **Volume**. This will enable you to input a diameter and a length and have HYSYS calculate the volume.
32. To obtain a volume of approximately 660 ft3, input a **Diameter** of 8 ft and a **Length** of 13 ft.
33. A horizontal unit will be used for the **Reboiler**. Ensure that **Horizontal cylinder** is selected for the **Level Calculator**.

Figure 8.20



Test the Model

At this point, the condenser and reboiler have both been sized and enough pressure-flow specifications have been set to reduce the degrees of freedom to zero. You can confirm that all the necessary pressure flow specifications have been added by selecting **Equation View Summary** from the **Simulation** menu.

34. Press the **Full Analysis** button to confirm that the case is properly defined for dynamic simulation.

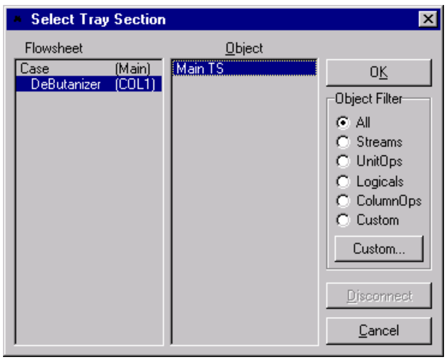
Tray Section

For the sizing of the tray section, a quick **Auto-Section** will be performed using the **Tray Sizing** utility. HYSYS will determine a tray diameter based on Glitsch design parameters for valve trays.

Using the Tray Sizing Utility

35. Use the hotkey combination **CTRL U** to open the **Utilities** view.
36. Select **Tray Sizing** from the list of utilities and press the **Add Utility** button.
37. On the **Tray Sizing** property view, press the **Select TS** button to attach the **DeButanizer** tray section.
38. Select **DeButanizer** from the **Flowsheet** group and **Main TS** from the **Object** group on the **Select Tray Section** dialog.

Figure 8.21



39. Click **OK**.
40. Click the **Auto Section** button on the **Tray Sizing** property view.

For this tray sizing analysis, all HYSYS defaults will be used. This will provide a good initial estimate of the tray diameter required for the **DeButanizer** tower.

41. Click the **Next** button on the **Auto Section Information** dialogue.

You can review the default design parameters on the **Tray Section Information** view. The parameters are grouped on the two tabs, **Setup** and **Sizing**.

42. Since all default design parameters will be used for this analysis, click the **Complete AutoSection** button.
43. On the **Results** page of the **Performance** tab, you will notice the **Section Pressure Drop** that HYSYS has calculated. This value is less than the original pressure profile input for the tower **DeButanizer**. You will modify this later.

Figure 8.22

Section	Section_1
Internals	Valve
Section Diameter [ft]	4.500
Max Flooding [%]	65.96
X-Sectional Area [ft2]	15.90
Section Height [ft]	30.00
Section DeltaP [psi]	1.011
Number of Flow Paths	1
Flow Length [in]	29.50
Flow Width [in]	51.18
Max DC Backup [%]	39.24
Max Weir Load [USGPM/ft]	90.31
Max DP/Tray [psi]	0.070
Tray Spacing [in]	24.00
Total Weir Length [in]	45.23
Weir Height [in]	2.000
Active Area [ft2]	10.48

44. Notice that HYSYS has calculated a section diameter of 4.5 feet.

Modifying the Tray Section Parameters

45. Press **F3** to open the **Object Navigator**.
46. In the **Flowsheets** group, select **DeButanizer**.
47. In the **Unit Operations** group, choose **Main TS**.
48. Press the **View** button.
49. On the **Main TS** property view, select the **Rating** tab, **Sizing** page.
50. Supply a tray **Diameter** of 4.5 feet in the **Tray Dimensions** group.

The pressure profile of the tray section will be modified to more closely match the results obtained from the tray sizing utility.

51. Open the **DeButanizer** property view and select the **Profiles** page of the **Parameters** tab.
52. Change the pressure in the **Reboiler** to 211 psia.
53. Change the pressure of **tray 15** to 209 psia.

8.6.3 Setting up the Controllers

Controllers will be added to the flowsheet to control the levels in the condenser and reboiler. Although these controllers are not required to run in dynamic mode, they will increase the realism and provide more model stability.

54. Add a **PID CONTROLLER** which will serve as the condenser level controller and specify the following details:

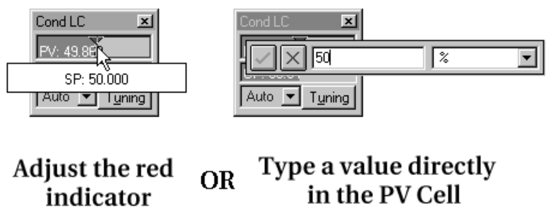
Tab [Page]	Input Area	Entry
Connections	Name	Cond LC
	Process Variable Source	Condenser@COL1 - Liquid Percent Level
	Output Target Object	Butanes@COL1
Parameters [Configuration]	Action	Direct
	Kc	2
	Ti	5 minutes
	PV Minimum	0%
	PV Maximum	100%

55. Press the **Face Plate** button.
56. Press the **Control Valve** button on the **Cond LC** property view.
57. In the **Valve Sizing** group specify the following:

Input Area	Entry
Flow Type	LiqVolFlow
Min Flow	0 ft3/hr
Max Flow	660 ft3/hr

58. Change the controller mode to **Auto** on the face plate.
59. Input a set point of 50%.

Figure 8.23



60. Add a **PID CONTROLLER** which will serve as the reboiler level controller and specify the following details:

Tab [Page]	Input Area	Entry
Connections	Name	Reb LC
	Process Variable Source	Reboiler@COL1 - Liquid Percent Level
	Output Target Object	C5+@COL1
Parameters [Configuration]	Action	Direct
	Kc	2
	Ti	5 minutes
	PV Minimum	0%
	PV Maximum	100%

61. Press the **Face Plate** button.
62. Press the **Control Valve** button on the **Cond LC** property view.
63. In the **Valve Sizing** group specify the following:

Input Area	Entry
Flow Type	LiqVolFlow
Min Flow	0 ft3/hr
Max Flow	750 ft3/hr

64. Change the controller mode to **Auto** on the face plate.
65. Input a set point of 50%.
66. You may have to re-initialize both controllers by switching the modes to Manual and then back to Automatic.
67. Use the **Save As** command from File Menu Bar to save the case as *debDYN.hsc*.

This case will be used as the starting point for the rating dynamics section.

8.6.4 Monitoring in Dynamics

Now that the model is ready to run in **Dynamic** mode, a strip chart can be created to monitor the general trends of key variables.

68. Open the Databook by using the hotkey combination **CTRL D**.
69. On the **Variables** tab, press the **Insert** button.
70. Add all of the variables that you would like to monitor.

A list of suggested variables is given below:

Variables To Manipulate	Variables To Monitor
Pressure of **Vent Control**	**Condenser** Liquid Percent Level
Liquid Volume Flowrate of **Feed 1**	**Reboiler** Liquid Percent Level
Liquid Volume Flowrate of **Feed 2**	Liquid Volume Flowrate of **Butanes**
SP for **Cond LC** and/or **Reb LC**	Liquid Volume Flowrate of **C5+**

71. After all variables have been added, close the Variable Navigator.
72. Select the **Strip Charts** tab from the Databook view.
73. Press the **Add** button.
74. Check the **Active** check box for each of the variables that you would like to monitor.
75. If required, add more strip charts.
76. Change the configuration of each strip chart by pressing the **Setup** button.
77. On the **Strip Charts** tab in the Databook, press the **Strip Chart** button to view each strip chart.
78. Start the integrator and allow the variables to line out.
79. Perform an analysis by manipulating variables and viewing the responses of the other variables.

9 Reactors

In **Dynamics** mode, the Continuously Stirred Tank Reactor (**CSTR**) and the General Reactors (**CONVERSION**, **EQUILIBRIUM** and **GIBBS** Reactors) all require similar specifications. This chapter will describe the **Rating** and **Dynamics** tabs for these similar reactors congruently.

The Plug Flow Reactor (**PFR**), requires quite different Ratings and Dynamic specifications. As a result, this reactor will be described in Section 9.2 - Plug Flow Reactor Dynamics.

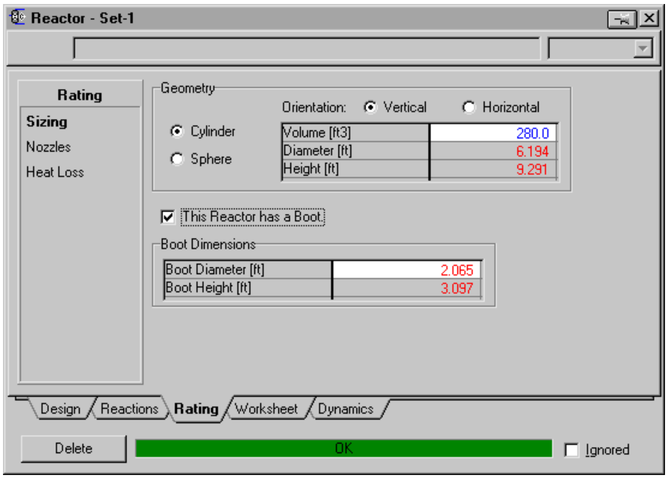
9.1 CSTR and General Reactors

9.1.1 Rating Tab

Sizing Page

On the **Sizing** page, you can define the geometry of the unit operation. It also allows for the specification of a **Boot**, if the vessel contains one.

Figure 9.1



The view consists of three main objects:

Object	Description
Geometry	Allows for the specification of the vessel geometry
This Reactor has a Boot	When activated, displays the **Boot Dimensions** group.
Boot Dimensions	Allows you to specify the dimensions of the vessel's boot.

Geometry Group

This group contains 5 objects which aid in the specification of the vessel volume:

Object	Description
Cylinder / Sphere	Toggles the shape of the vessel between Sphere and Cylinder. This will affect the number of specifications required as well as the method of volume calculation. If **Cylinder** is selected and a diameter and height have been specified, the vessel volume will be calculated as: $$V_{reactor} = \left(\frac{Diameter^2}{4}\pi \times Height\right) + V_{boot} \quad (9.1)$$ If **Sphere** is selected and either the height **or** diameter have been specified, the vessel volume is calculated as: $$V_{reactor} = \frac{(Height\ or\ Diameter)^3\pi}{6} + V_{boot} \quad (9.2)$$ *where:* $V_{reactor}$ *= the Volume of the reactor* V_{boot} *= the volume of the boot* *Height, Diameter = the values taken from the respective fields*
Orientation	You may choose the orientation of the vessel as either: • **Horizontal -** the ends of the vessel 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 value in this field: • If the height and/or diameter have been entered, this field will display the value calculated using either **Equation (9.1)** or **Equation (9.2)**. • If you enter a value into this field and either the Height (Length) or Diameter is specified, HYSYS will back calculate the other parameter using **Equation (9.1)** for cylindrical vessels. For spherical vessels, Height and Diameter will be back calculated as the Height will be equal to the Diameter. • If you enter a value into this field (and only this field) both the **Height (Length)** and **Diameter** will be calculated assuming a ratio of 3:2 (i.e. Height:Diameter).
Diameter	Holds the diameter of the vessel. If the vessel is a Sphere, then the Diameter will be the same as the Height (Length).
Height / Length	Holds the height or length of a **Cylindrical** vessel depending on the vessel's orientation (horizontal or vertical). If the vessel is a **Sphere**, the Height/Length will be the same as the Diameter.

Boot Dimensions

If the reactor you are rating has a Boot, you may include its volume in the total vessel volume by activating the **This Reactor has a Boot** check box. This will make the **Boot Dimensions** group box visible. The **Boot Dimensions** group consists of two fields:

Field	Description
Boot Diameter	This is the diameter of the boot. The default value for this field is usually one third of the reactor diameter.
Boot Height	This is the height of the boot. The default value for this field is half the reactor diameter.

The volume of the boot is calculated using a cylindrical volume calculation:

$$V_{Boot} = \pi\left(\frac{Boot\ Diameter}{2}\right)^2 \times Boot\ Height \tag{9.3}$$

Therefore the default boot volume is:

$$\begin{aligned} V_{Boot} &= \pi\left(\frac{Diameter}{6}\right)^2 \times \frac{Diameter}{2} \\ &= \frac{\pi(Diameter)^3}{72} \end{aligned} \tag{9.4}$$

where: Diameter = the Reactor Diameter

This means that the Boot Volume can be estimated by entering the total Reactor Volume.

Nozzles Page

The **Nozzles** page contains information regarding the elevation and diameter of the nozzles. The Plant+ license is required to use the Nozzle features found on this page. Refer to **Section 1.5 - Plant+** for more information.

Unlike steady state vessel operations, the placement of feed and product nozzles on a dynamic reactor 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 will draw material from the liquid holdup. If the product nozzle is located above the liquid level, the exit stream will draw 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 nozzle the liquid is.

Essentially, all vessel operations in HYSYS are treated similarly. The composition and phase fractions (i.e. 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. Therefore, 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

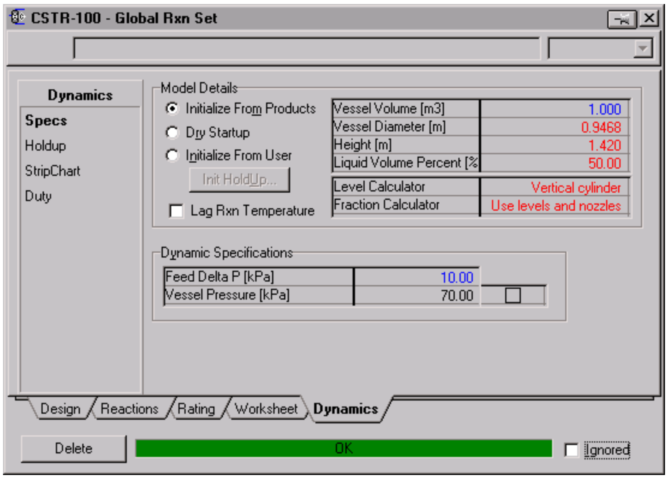
The **Heat Loss** page contains heat loss parameters which characterize the amount of heat lost across the vessel wall. For more information about the Heat Loss page see **Section - Heat Loss Page** in **Chapter 7 - Separation Operations**

9.1.2 Dynamics Tab

Specs Page

The **Specs** page contains information regarding initialization modes, vessel geometry, and vessel dynamic specifications:

Figure 9.2



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.

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 **Liquid Volume Percent** cell.

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 **Liquid Volume Percent** cell 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 pressing the ***Init Holdup*** button. The liquid level is set to the value indicated in the **Liq Volume Percent** cell.

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 check the Lag Rxn Temperature box, the reaction solver flashes with the explicit Euler method. Otherwise, for a dynamic run, the steady-state reaction solver will always flash with the implicit Euler methods which could be extremely 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.

The vessel geometry can be specified in the **Model Details** section. The following vessel geometry parameters can be specified in the same manner as the **Geometry** section in **Sizing** page of the **Ratings** tab:

- Vessel Volume
- Vessel Diameter
- Vessel Height (Length)
- (Vessel Geometry) Level Calculator

Liquid Volume Percent

The **Liquid Level Percent** is displayed in this section. You can modify the level in the vessel at any time. HYSYS will then use this level as an initial value when the Integrator is started.

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 following is a description of each Fraction Calculator option:

Fraction Calculator Mode	Description
Use Levels and Nozzles	The nozzle location and vessel liquid level affect the product composition as detailed in **Nozzles**, **Section 9.1.1 - Rating Tab**.
Emulsion Liquid	This setting is used only in tray section holdups. It should not be used anywhere else.
Trivial	This setting is used only in unit operations with negligible holdup such as a valve or pump operation. It should not be used anywhere else.

The **Fraction Calculator** defaults to the correct mode for all unit operations and does not typically require changing.

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** cell. The frictional pressure losses at each product nozzle are automatically set to zero by HYSYS.

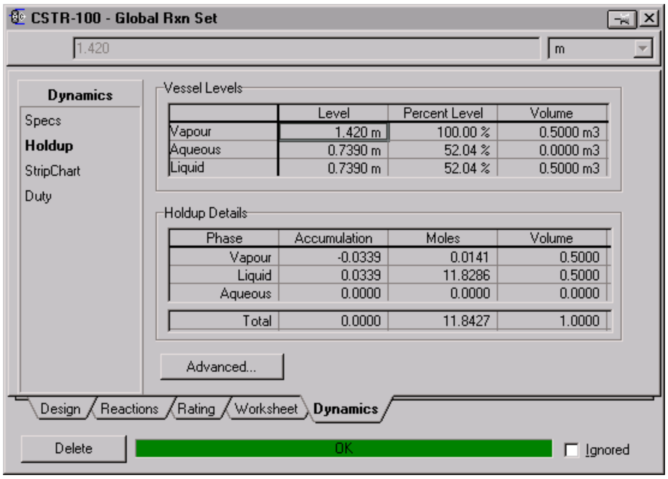
It is recommended that you enter a value of zero in the **Feed Delta P** cell since a fixed pressure drop in the vessel is not realistic for all flows. If you wish to model friction loss at the inlet and exit stream, it is suggested you add **Valve** operations. In this case, flow into and out of the vessel will be realistically modelled.

The vessel pressure can also be specified. This specification can be made active by checking the box beside the **Vessel Pressure** cell. This specification is typically not set since the pressure of the vessel is usually variable and determined from the surrounding pieces of equipment.

Holdup Page

The **Holdup** page contains information regarding the holdup's properties, composition, and amount.

Figure 9.3



Vessel Levels Group

For each possible phase in the vessel (vapour, aqueous and liquid), the **Vessel Levels** group will display:

Vessel Levels	Description
Level	The cumulative height of the phase.
Percentage Level	The cumulative height of the phase as percentage of the vessel height.
Volume	Total volume of the phase.

Holdup Details Group

For each phase contained within the volume space of the unit operation, the following is specified:

Holdup Details	Description
Phase	The phase of the holdup volume.
Accumulation	The accumulation refers to the rate of change of material in the holdup for each phase.

Holdup Details	Description
Moles	Indicate the amount of material in the holdup for each phase.
Volume	Indicate the holdup volume of each phase.

Click the Advanced button to accesses a view that provides more detailed information about the holdup of that unit operation. Refer to Section 1.2.7 - Advanced Holdup Properties for more information.

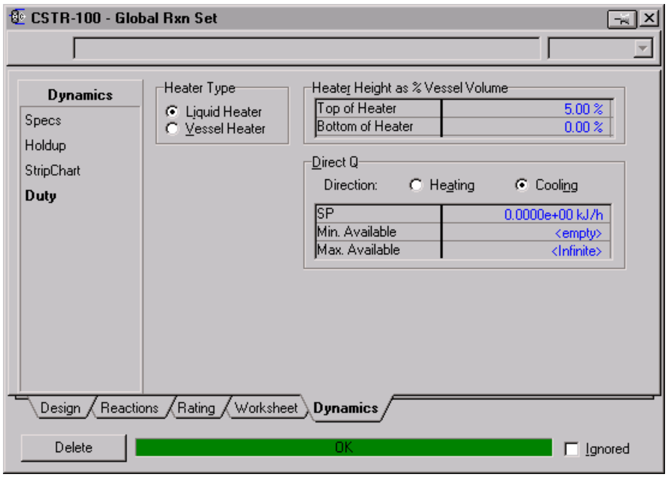
Stripchart Page

The **Stripchart** page allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the **Create Stripchart** button to display the stripchart.

Duty Page

The **Duty** page opens a list of available heating methods for the vessel operation.

Figure 9.4



Duty Source

You can choose whether HYSYS calculates the duty applied to the vessel from a **Direct Q** option or a **Utility Fluid** option by selecting the corresponding radio button.

If the **Direct Q** source is chosen, you may directly specify the duty applied to the holdup in the setpoint (**SP**) cell. If the **Utility Fluid** source is chosen, you may 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 **Heat Flow** cell. For more information regarding how the utility fluid option calculates duty, see **Chapter 10 - Logical Operations**.

If the **Heating** radio button is selected, the duty shown in the **SP** or **Heat Flow** fields will be added to the holdup. If the **Cooling** radio button is selected the duty will be subtracted from the holdup.

Heater Type

There are two heating methods available to the general vessel operation:

If a **Vessel Heater** is chosen, 100% of the duty specified or calculated in the **SP** cell is applied to the vessel's holdup:

$$Q = Q_{\text{Total}} \tag{9.5}$$

where: Q *= the total heat applied to the holdup*

Q_{Total} *= the duty calculated from the Duty Source*

If a **Liquid Heater** is chosen, the duty applied to the vessel depends on the liquid level in the tank. The heater height must be specified if the **Liquid Heater** is chosen. The heater height is expressed as a percentage of the vessel volume. 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.

$$\begin{aligned} Q &= 0 && (L < B) \\ Q &= \frac{L-B}{T-B} Q_{Total} && (B \le L \le T) \\ Q &= Q_{Total} && (L > T) \end{aligned} \tag{9.6}$$

where: L *= Liquid Percent Level (%)*

T *= Top of Heater (%)*

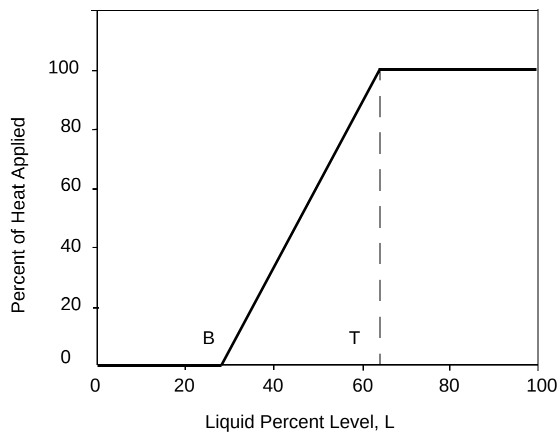
B *= Bottom of Heater (%)*

The Percent Heat Applied may be calculated as follows:

$$Percent\ Heat\ Applied = \frac{Q}{Q_{Total}} \times 100\% \tag{9.7}$$

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 9.5



Percent Heat Applied for a Liquid Heater

9.1.3 CSTR Dynamics Example

In this example, a simple CSTR will be modelled in dynamics.

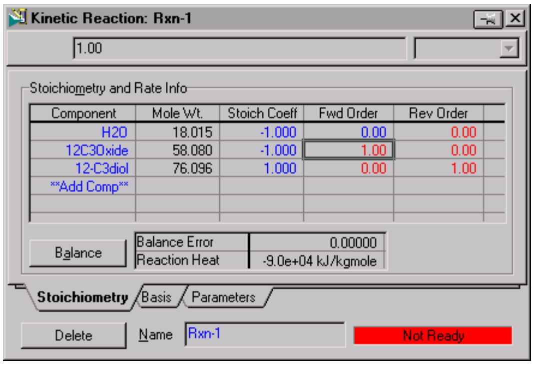
1. Begin by creating a new case.
2. Create the following Fluid Package:

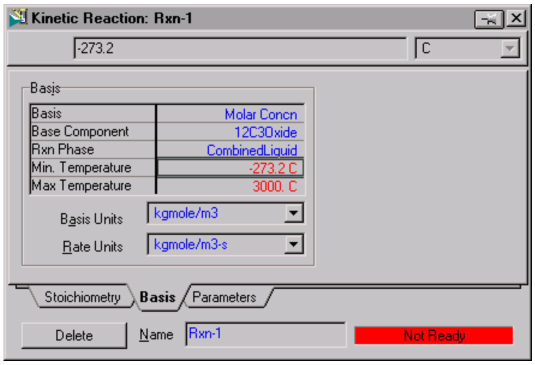
Property Package	Components
UNIQUAC	H2O, 12C3Oxide (Propylene Oxide), 12-C3diol (Propylene Glycol) and Nitrogen.

3. On the **Binary Coeffs** tab, estimate the unknown binary coefficients using the **UNIFAC VLE** coefficient estimation method.
4. Go to the **Reactions** tab of the Basis Manager and select the **Add Comps** button. In the **Reaction Components Selection** view select the **FPkg Pool** radio button and click the **Add this Group of Components** button.

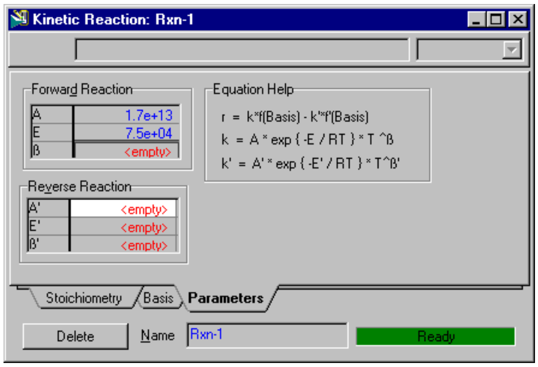
5. Add a **Kinetic Reaction**. Specify the **Kinetic Reaction** property view as show in **Figure 9.6** :

Figure 9.6





1. On the **Stoichiometry** tab, you must specify the components and their stoichiometric coefficients.
2. On the **Basis** tab, you must set the **Base Component** and Reaction Phase (**Rxn Phase**).
3. On the **Parameters** tab, set the rate constant parameters.



6. Add the Global Reaction Set to the Fluid Package you created earlier and enter the Simulation Environment.
7. Add the following Material streams to the Flowsheet:

Stream Name	Feed	Make Up
Temperature [C]	25	60
Pressure [kPa]	130	130
Molar Flow [kgmole/hr]	350	0
Comp Mole Frac [H2O]	0.8	0
Comp Mole Frac [12C3Oxide]	0.2	0
Comp Mole Frac [12-C3diol]	0	0
Comp Mole Frac [Nitrogen]	0	1

You can add a CSTR by either selecting the ***CSTR*** *button on the Object Palette or press the* ***F12*** *and from the Unit Ops dialogue select the* ***Cont. Stirred Tank Reactor****.*

8. Now install a CSTR Reactor with the following specifications:

UNIT OPERATION TYPE	CONTINUOUSLY STIRRED TANK REACTOR	
Tab [Page]	**Input Area**	**Entry**
Design [Connections]	Name	CSTR-100
	Feeds	Feed Make Up
	Liquid Outlet	Reactor Out
	Energy	Coolant
Design [Parameters]	Delta P	0
	Duty	0 kJ/hr Cooling
	Liquid Level	50
	Volume	8 m^3
Reactions [Details]	Reactions Set	Global Rxn Set

9. Install a **VALVE** operation with the following connections:

UNIT OPERATION TYPE	VALVE	
Tab [Page]	**Input Area**	**Entry**
Design [Connections]	Name	VLV-100
	Feed	Reactor Out
	Product	Product
Design [Parameters]	Delta P	68 kPa

10. Switch to **Dynamics** mode.
11. Press the **Dynamics Assistant** button.

Dynamics Mode Button

Dynamic Assistant Button

A list of modifications to the Flowsheet will be displayed in order to run the case in **Dynamics** mode. Note that the Dynamic Assistant presents suggestions for setting up dynamic simulations. Its suggestions may not always be compatible with the system you are trying to model. For this model, the Assistant will make suggestions about un-initialized streams as well as sizing the valve. For this case all of the suggestions are useful.

12. Press the **Make Changes** button in order to size the valve and initialize the streams.
13. As mentioned earlier, the **Make Up** stream will be used to maintain the pressure of the reactor. Open the **Make Up** stream property view and go to the **Dynamics** tab. Now **disable** the flow specification of this stream.

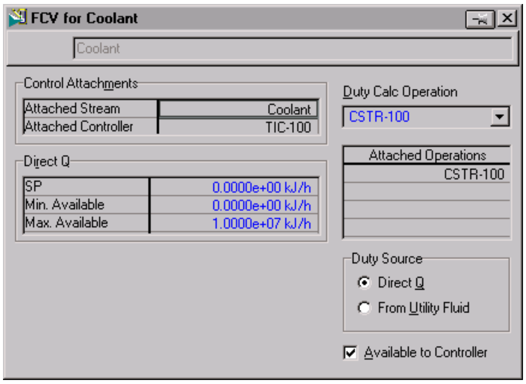
You require a Plant+ license in order to use the Nozzles page. If you have a Plant+ license but can not see the Nozzles page refer to ***Section 1.5 - Plant+*** *for information on activating the Plant+ features. Otherwise skip this step.*

14. Go to the **Nozzles** page in the reactor's **Ratings** tab. Ensure that the nozzle Elevation Parameters for each stream, appear as shown below:

Elevation Parameter	Feed	Make Up
Elevation [% Height]	85%	100%

15. In order to maintain the reactor temperature at 60°C, a controller must be attached to the **Coolant** stream. Before you add the Controller, it is suggested you first size the Control Valve. Open the **Coolant** stream property view. On the **Dynamics** tab, click the **Utility Valve** button and specify the information seen in the **Direct Q** group box.

Figure 9.7



On the ***Parameters*** *tab, you* ***must*** *specify the PV Range before setting the* ***Mode*** *or* ***SP****.*

16. Add the following controllers:

UNIT OPERATION TYPE	CONTROLLER	
Tab [Page]	**Input Area**	**Entry**
Connections	PV Source Object	Reactor Out
	PV Source Variable	Temperature
	Output Target Object	Coolant
Parameters [Configuration]	Action	Direct
	Mode	Auto
	SP	60 °C
	Kc	5
	Ti	1
	PV Minimum	20 °C
	PV Maximum	150 °C

On the ***Parameters*** *tab, you* ***must*** *specify the PV Range before setting the* ***Mode*** *or* ***SP****.*

UNIT OPERATION TYPE	CONTROLLER	
Tab [Page]	**Input Area**	**Entry**
Connections	PV Source Object	CSTR-100
	PV Source Variable	Liquid Percent Level
	Output Target Object	VLV-100
Parameters [Configuration]	Action	Direct
	Mode	Auto
	SP	85
	Kc	2
	Ti	10
	PV Minimum	0%
	PV Maximum	100%

17. In the DataBook, select the following variables:

Object	Variable	Variable Specifics
Make Up	Molar Flow	
Coolant	Heat Flow	
Product	Comp Mole Frac	12-C3diol
Product	Comp Mole Frac	Nitrogen
Product	Molar Flow	
CSTR-100	Liquid Percent Level	
Reactor Out	Temperature	

18. Add a Strip Chart and attach the variables specified above to the chart.
19. Start the Integrator.

As the Dynamic simulation continues, you will see the all the variables in the strip chart begin to level out. Eventually all the variables will reach their steady state values.

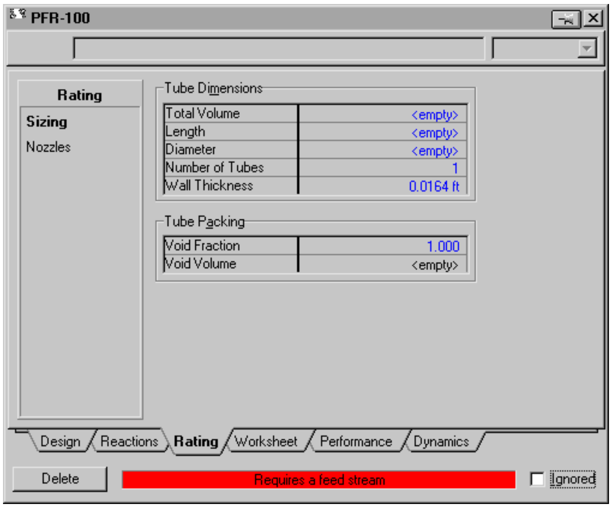
9.2 Plug Flow Reactor Dynamics

9.2.1 Rating Tab

The **Rating** tab consists of two pages: **Sizing** and **Nozzles**.

Sizing Page

Figure 9.8



On the **Sizing** page, you can specify the **Tube Dimensions** and the **Tube Packing** information in their respective groups.

Tube Dimensions

For the tube dimensions, you will need to specify any three of the following four parameters:

Tube Dimension	Description
Total Volume	The total volume of the PFR.
Length	The total length of the of an individual tube.

Tube Dimension	Description
Diameter	The diameter of an individual tube.
Number of tubes.	The total number of tubes required. This will always be calculated to the nearest integer value.

When three of these dimensions are specified, the fourth will automatically be calculated. Note that the **Total Volume** refers to the combined volume of all tubes.

By default, the number of tubes is set to 1. Although the number of tubes is generally specified, you may set this parameter as a calculated value by selecting the **Number of Tubes** field and pressing the DELETE key. The number of tubes will always be 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 will have to re-specify the tube dimensions.

The **Tube Wall Thickness** may also be specified.

The void volume is used to calculate the spatial velocity, which impacts the rate of reaction.

Tube Packing

This group consists of two fields: **Void Fraction** and **Void Volume**. The **Void Fraction** is by default set to 1, in which case there is no catalyst present in the reactor. The resulting **Void Volume** will be 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 will also be 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

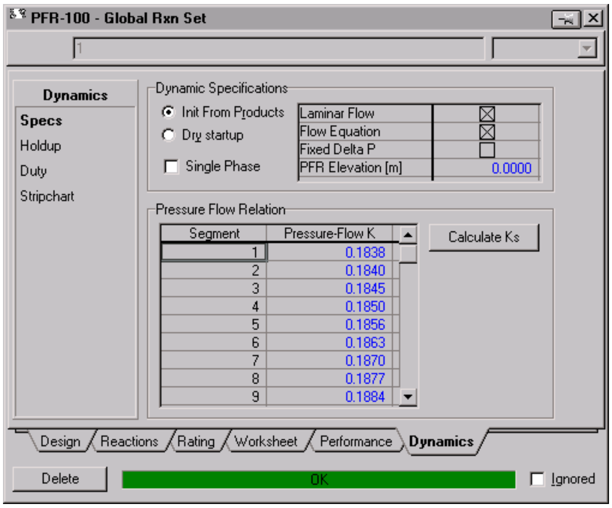
The **Nozzles** page contains information regarding the elevation and diameter of the nozzles. The Plant+ license is required to use the Nozzle features found on this page. Refer to **Section 1.5 - Plant+** for more information.

9.2.2 Dynamics Tab

The **Dynamics** tab contains four pages: **Specs**, **Holdup**, **Duty** and **Stripchart**.

Specs Page

Figure 9.9



Dynamic Specifications Group

The **Dynamic Specifications** group consists of 7 objects:

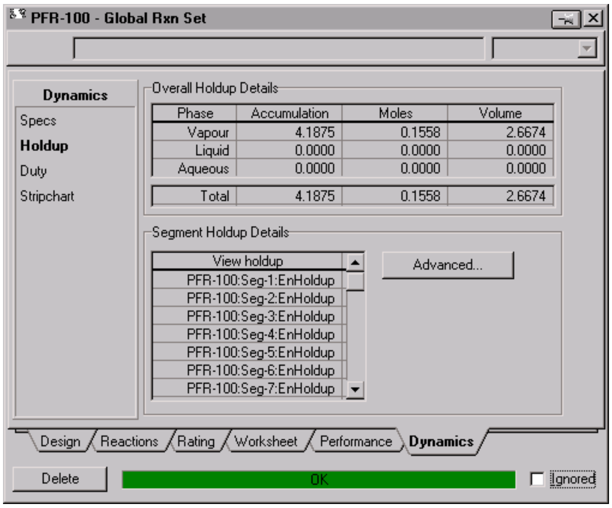
Objects	Description
Initialize from Products	The composition of the holdup is calculated from a weighted average of all products exiting the holdup. A PT flash is performed to determine other holdup conditions.
Dry Startup	The composition of the holdup is calculated from a weighted average of all feeds entering the holdup. A PT flash is performed to determine other holdup conditions.
Laminar Flow	Assumes laminar flow in the PFR.
Flow Equation	Uses the flow equation to calculate the pressure gradient across the PFR. You are required to either estimate k values in steady-state (by pressing the **Calculate K's** button) or providing your own values in **Pressure Flow Relation** group.
Fixed Delta P	Assumes a constant pressure drop across the PFR. Does not require k values.
PFR Elevation	The height above ground that the PFR is currently positioned.

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 may enter your own k values into this table or, while you are in **Steady State** mode, you can press the **Calculate K's** button and HYSYS will calculate the k values using the steady state data.

Holdup Page

Figure 9.10



The **Holdup** page contains information regarding the properties, composition and amount of the holdup in each phase in the PFR. The page consists of two groups: **Overall Holdup Details** and **Segment Holdup Details**.

Overall Holdup Details Group

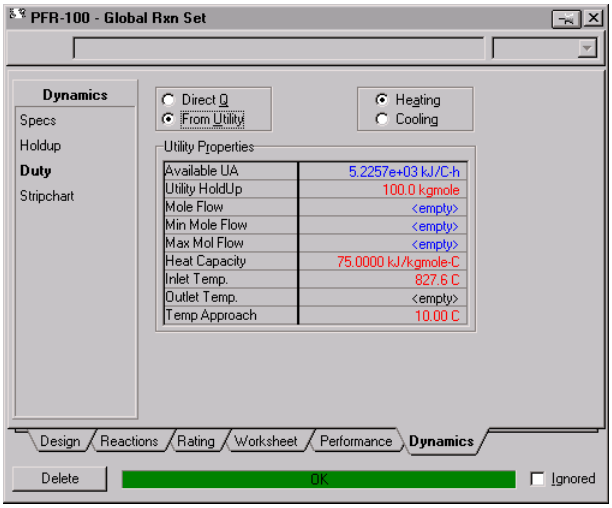
For each phase contained within the volume space of the **PFR**, the following is displayed:

Holdup Details	Description
Phase	The phase of the holdup volume.
Accumulation	The accumulation refers to the rate of change of material in the holdup for each phase.
Moles	Indicate the amount of material in the holdup for each phase.
Volume	Indicate the holdup volume of each phase.

Click the Advanced button to accesses a view that provides more detailed information about the holdup of that unit operation. Refer to Section 1.2.7 - Advanced Holdup Properties for more information.

Duty Page

Figure 9.11



You can choose whether HYSYS calculates the duty applied to the vessel from a **Direct Q** option or a **Utility Fluid** option by selecting the corresponding radio button.

If the **Direct Q** source is chosen, you may directly specify the duty applied to the holdup in the **SP** cell. If the **Utility Fluid** source is chosen, you may 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 **Heat Flow** cell. For more information regarding how the utility fluid option calculates duty, see **Chapter 10 - Logical Operations**.

If the **Heating** radio button is selected, the duty shown in the **SP** field will be added to the holdup. If the **Cooling** radio button is selected the duty will be subtracted from the holdup.

Stripchart Page

The **Stripchart** page allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the **Create Stripchart** button to display the stripchart.

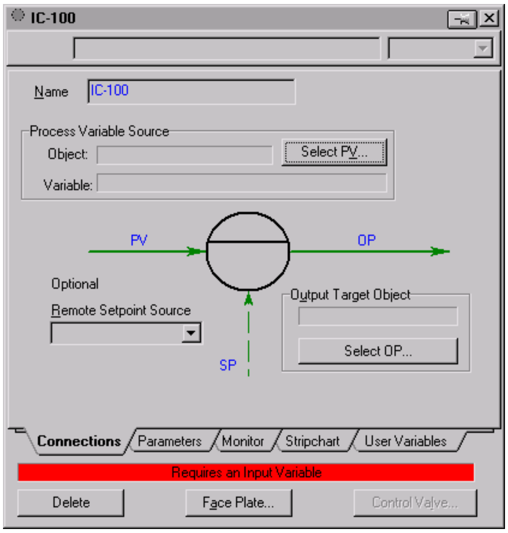
10 Logical Operations

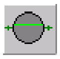
10.1 PID Controller

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 10.1





PID Controller Button

To install the **Controller** operation, choose **Add Operation** from the **Flowsheet** menu, and select **PID Controller**. Alternatively, select the PID Controller button in the Object Palette.

10.1.1 Connections Tab

The **Connections** tab allows you to select both the PV and OP. It is comprised of six objects:

Object	Description
Name	This field contains the name of the controller. It can be edited by selecting the field and entering the new name in the Edit Bar.
Process Variable Source Object	This field contains the Process Variable Object (stream or operation) that owns the variable you wish to control. It is specified via the Variable Navigator.
Process Variable	This field contains the Process Variable you wish to control.
Output Target Object	The Output Target Object field is 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

Process Variable Source

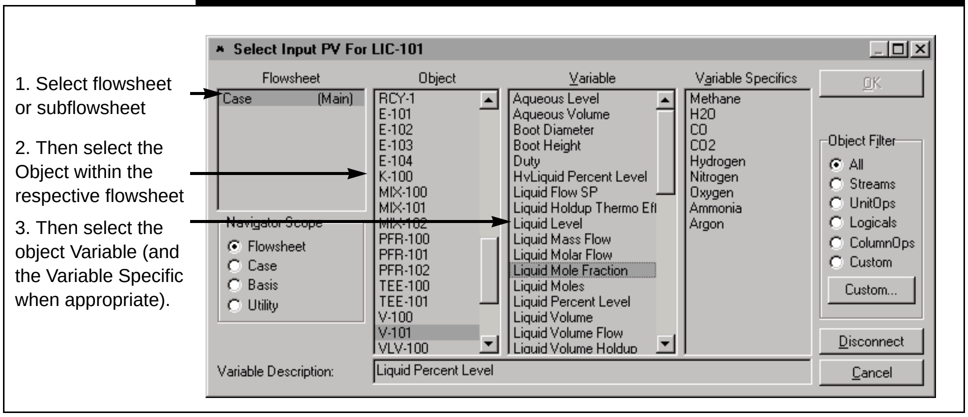
The Process Variable, or PV, is the variable that must be maintained or controlled at a desired value.

Common examples of PVs include vessel pressure and liquid level, as well as stream conditions such as flow rate or temperature.

The Variable Navigator allows you to simultaneously select the Object and Variable.

To attach the **Process Variable Source**, click the Select PV button. Then select the appropriate object and variable simultaneously, using the **Variable Navigator** (**Figure 10.2**). For more information, see **Section 5.2 - Navigation** of the User's Guide.

Figure 10.2



Remote Setpoint Source

A Spreadsheet cell can also be a Remote Setpoint Source.

The "cascade" mode of the controller no longer exits. Instead what we have 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, i.e., a master in the classical cascade control scheme.

The **Remote Setpoint Source** drop down box allows the user to select the remote sources from a list of existing operations.

Output Target Object

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** view.

The Controller compares the Process Variable to the Setpoint and produces an output signal which causes the control valve to open or close appropriately.

Selecting the **Output Target Object** is done in a similar manner to selecting the **Process Variable Source**. In this case, however, you are only selecting the Object. Only Objects with Control Valves not currently attached to another Controller will be accessible.

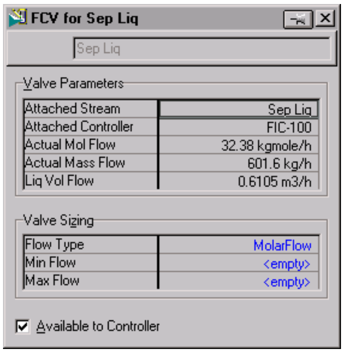
The information regarding the Valve sizing is contained on a Sub-View accessed via the Control Valve button found at the bottom of the **PID Controller** property view.

10.1.2 Control Valve

The information shown on the **Control Valve** view is specific to the associated valve. For instance, the information for a Vapour Valve is different than that for an Energy Stream.

FCV for a Liquid/Vapour Product Stream from a Vessel

Figure 10.3



The **FCV** view for a material stream consists of two group boxes: **Valve Parameters** and **Valve Sizing.** 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 view that requires specification. The group box contains three fields:

Field	Description
Flow Type	The type of flow you wish to specify: molar flow, mass flow, liquid volume flow, or 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):

$$Flow = \frac{OP(\%)}{100}(Maximum - Minimum) + Minimum \qquad (10.1)$$

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

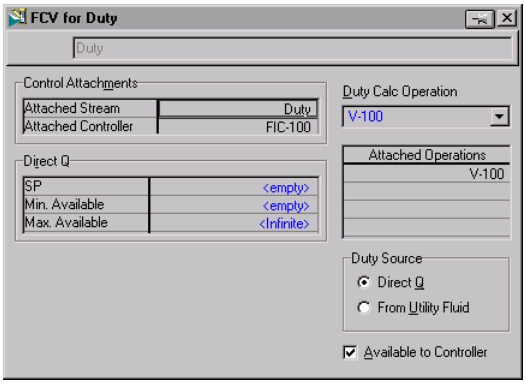
The **FCV** view that will appear will be dependent on the type of duty stream selected. There are two types of duty streams: the **Direct Q** duty that consists of a simple power value (i.e. BTU) and the **Utility Fluid** which takes the duty from a utility fluid (i.e. steam) with known properties.

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** (see **Duty Source** group box).

Figure 10.4

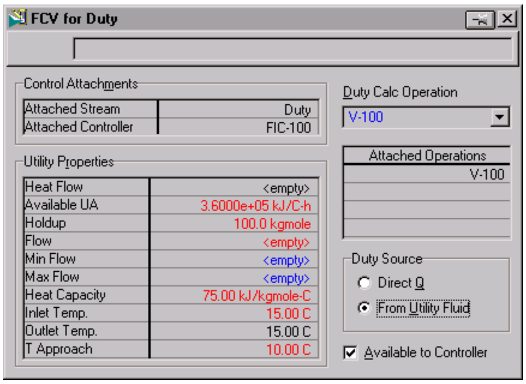


The Attached Stream and Controller are displayed in the upper left corner of the view in the **Control Attachments** group. The specification required by the view are all entered into the **Direct Q** group box. In this group, Setpoint (**SP**) is displayed, and you may supply the minimum (**Min. Available**) 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 are displayed in the upper left corner of the view.

Figure 10.5



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	This is the operation outlet temperature minus the outlet temperature of the Utility Fluid.

Available to Controller Check Box

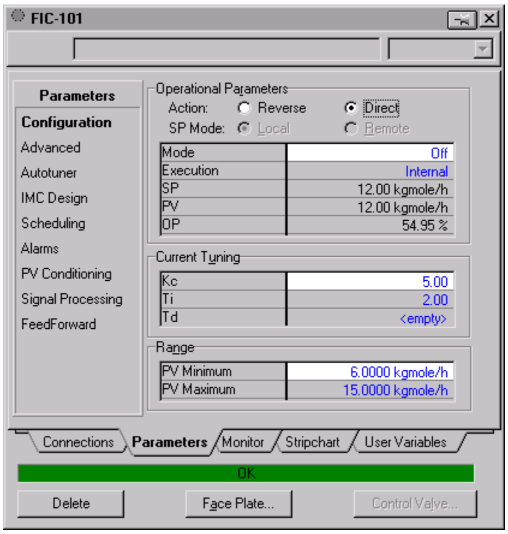
When you make the controller connections and move to the **Control Valve** view (by clicking the Control Valve button on the **PID Controller** property view), the **Available to Controller** check box will automatically be checked. HYSYS assumes that because you installed a new controller on the valve, you probably wish to make it available to the Controller.

10.1.3 Parameters Tab

The **Parameters** tab consists of nine pages: **Configuration**, **Advanced**, **Autotuner**, **IMC Design**, **Scheduling**, **Alarms**, **PV Conditioning**, **Signal Processing** and **FeedForward**.

Configuration Page

Figure 10.6



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.

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 input by the user or displayed only.

For the Controller to become operational, you must:

Note that without a PV span, the Controller will not function.

1. Define the minimum and maximum values for the PV (the Controller will not 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. Note that HYSYS will use the current value of the PV as the set point by default, but you may change this value at any time.
3. 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:

$$PV(\%) = \left(\frac{PV - PV_{min}}{PV_{max} - PV_{min}}\right)100 \qquad (10.2)$$

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 will become whatever the PV is at the particular valve opening you specify.

Modes

The **Controller** will operate 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	Manipulate the Controller output manually.

Controller Mode	Description
Auto	The Controller reacts to fluctuations in the Process Variable and manipulates the Output according to the logic defined by the tuning parameters.
Indicator	This allows you to simulate the controller without controlling the process.

The mode of the controller may also be set on the Faceplate (see Section 10.2 - Digital Point for details).

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:

$$OP(t) = OP_{ss} + K_c E(t) + \frac{K_c}{T_i}\int E(t)\mathrm{d}t + K_c T_d \frac{dE(t)}{dt} \qquad (10.3)$$

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:

$$E(t) = SP(t) - PV(t) \qquad (10.4)$$

Depending on which of the three tuning parameters you have supplied, the Controller will respond accordingly to the Error. A Proportional-only controller is modelled 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:

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 wish 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**:

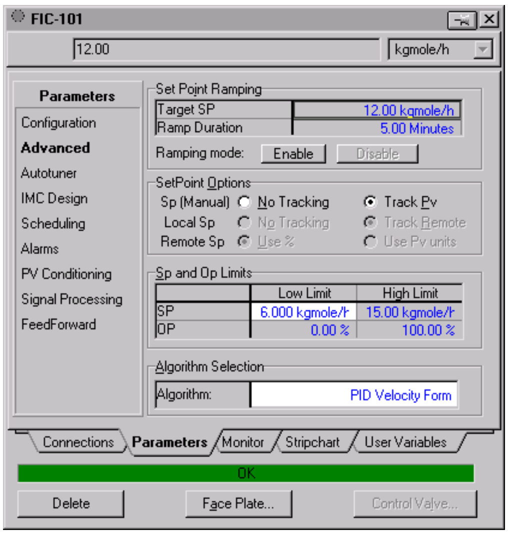
Process Variable and Setpoint	R-100 Stage Pressure
Output	RebDuty Flow
When R-100 Stage Pressure rises above the SP	The OP decreases, in effect decreasing the flow of RebDuty and decreasing the Stage Pressure of R-100.
When R-100 Stage Pressure falls below the SP	The OP increases, in effect increasing the flow of RebDuty and increasing the Stage Pressure of R-100.

SP Mode

You have now 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 the user can manually specify the setpoint via the view (interface), and the other is the remote setpoint which comes from another object such as a spread-sheet or another controller cascading down a setpoint, i.e., a master in the classical cascade control scheme.

Advanced Page

Figure 10.7



The second page of the **Parameters** tab is the **Advanced** page which contains the following four groups:

Group	Description
Set Point Ramping	This group allow you to specify the ramp target and duration.
SetPoint Options	This group contains the options for setpoint tracking.
Sp and Op Limits	This groups allows you to set the upper and lower limits for set point and output targets.
Algorithm Selection	This group contains the PID controller algorithms for output calculation.

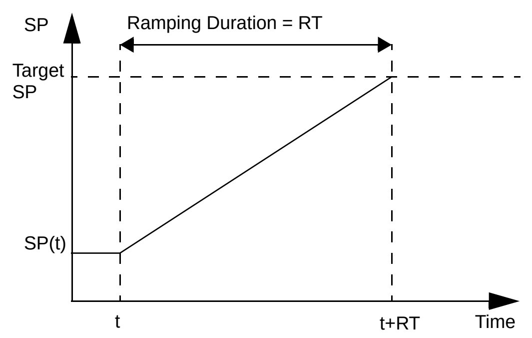
Setpoint Ramping

Setpoint ramping is only available in Auto mode.

The setpoint ramping function has been modified in the present PID controllers. Now it is continuous, i.e., when enabled by clicking the **Enable** button, the setpoint will change over the specified period of time in a linear manner. The Set Point Ramping group contains the following two fields:

Field	Input Required
Target SP	This field contains the Setpoint you wish the Controller to have at the end of the ramping interval. When the ramping is disabled, the **Target SP** field display the same value as the **SP** field on the **Configuration** page.
Ramp Duration	This field contains the time interval you wish to complete setpoint change in.

Figure 10.8



Controller Ramping

There are also two buttons available in this group:

Button	Functionality
Enable	Pressing this button activates the ramping process.
Disable	Pressing this button 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, will show the final value of the setpoint whereas the **SP** field, on the **Configuration** page, will show the current setpoint seen internally by the control algorithm. Note that during ramping, if a second setpoint change has been activated, then Ramping Duration time would be restarted for the new setpoint.

Example

If you click the **Enable** button and enter values for the two parameters in the **Set Point Ramping** group, the Controller will switch to Ramping mode and adjust the Setpoint linearly (to the **Target SP**) during the **Ramp Duration**, see **Figure 10.8**. For example, suppose your current SP is 100 and you wish 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 will adjust the SP from 100 to 150 linearly over the 10 minute interval.

SetPoint Options

In the past the PID controllers implemented an automatic setpoint tracking in manual mode, i.e., 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 you the Pv tracking, by activating 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 will be an instantaneous bumpless transfer prior to the controller recognizing a setpoint that is different from the Pv. If the **Track PV** radio button is active 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 activating the **Track Remote** button.

The **Remote Sp** option allows activate 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 %

Is this radio button is active then the controller would reads in a value in percentage from a remote source and use 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 a the new setpoint. The remote sources setpoint must have the same units as the controller Pv.

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 will internally convert the new setpoint as m^3/s i.e. 1/3600 = 0.00028 m^3/s and pass it to the controller, which will convert it back into m^3/h i.e. 1 m^3/h. The controller will use this value as the new setpoint. If the units are not specified, then the spreadsheet will pass it as 1 m^3/s, which is the base unit in HYSYS, and the controller will convert 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 will use the maximum value i.e. 1.75m^3/h as the new setpoint.

Sp and Op Limits

This group enables you to specify the output and setpoint limits. The output limits will ensure that a predetermined minimum or maximum output value is never exceeded. In the case of the setpoint, the limits will enforce an acceptable the range of values that could be entered via the interface or from a remote source.

Algorithm Selection

In the **Algorithm Selection** group you can select one of three available controller update algorithms:

ARW = Anti-Reset Windup

- PID Velocity Form
- PID Positional Form (ARW)
- PID Positional Form (noARW)

Velocity or Differential Form

In the velocity or differential form the controller equation is given as:

$$u(t) = u(t-1) + K_c\left[e(t) - e(t) + \frac{1}{T_i}e(kh) + T_d\frac{(e(t) - 2e(t-1) + e(t-2))}{h}\right] \qquad (10.5)$$

where: u(t) = the controller output and t is the enumerated sampling instance in time

u(t-1) = the value of the output one sampling period ago

Kc, Ti, and Td = the controller parameters

h = the sampling period

It is important to note that 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:

$$u(t) = K_c\left[e(t) - e(t) + \frac{1}{T_i}\sum_{k=0}^{n} e(kh) + T_d\frac{(e(t) - e(t-1))}{h}\right] \qquad (10.6)$$

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 will be 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 will respond 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.

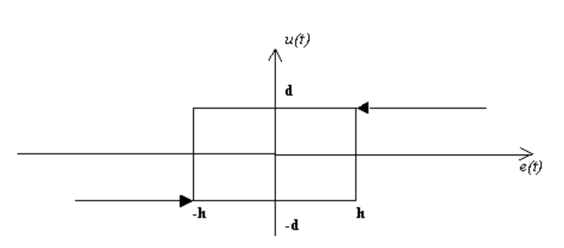
In HYSYS both algorithms are implemented as presented above with one key exception, there is no derivative kick. This means that the derivative part of the control algorithm operates on the process variable as opposed to the error term. As such the control equation given in **Equation (10.5)** is implemented as follows:

$$u(t) = u(t-1) + K_c\left[e(t) - e(t) + \frac{1}{T_i}e(kh) + T_d\frac{(-pv + 2pv(t-1) - pv(t-2))}{h}\right] \qquad (10.7)$$

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). **Figure 10.9** 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 10.9

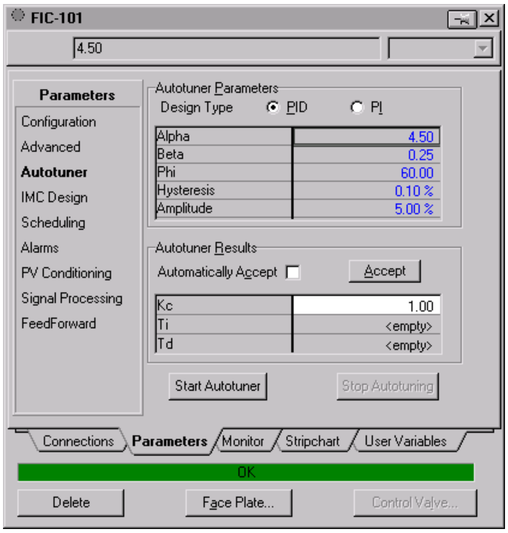


This type of relay is a double-valued nonlinearity, sometimes referred to as having memory, i.e., the value of the output depends on the direction that the process error is coming. Relays are quite common in automation and control, and this technique for tuning PID controllers has been around at least 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.

This tab allows you to specify the autotuning parameters.

Figure 10.10



The page contains two groups:

Group	Description
Autotuner Parameters	This group contains the parameters required by the Autotuner to calculate the controller parameters.
Autotuner Results	This group display the resulting controller parameters. You have the option to accept the results as the current tuning parameters.

Autotuner Parameters

In the present version of the software there are default values specified for the PID tuning. Before starting the autotuner the 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 will display the parameters range.

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 supply which are as follows:

Parameter	Range
Ratio (Ti/Td) (Alpha)	$3.0 \leq \alpha \leq 6.0$
Gain ratio (Beta)	$0.10 \leq \beta \leq 1.0$
Phase angle (Phi)	$30° \leq \phi \leq 65°$
Relay hysteresis (h)	$0.01\% \leq h \leq 5.0\%$
Relay amplitude (d)	$0.5\% \leq d \leq 10.0\%$

Autotuner Results

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 accepts the results either automatically or manually. Activating the **Automatically Accept** check box will set the resulting controller parameters as the current value instantly. If the **Automatically Accept** check box is not activated, you can specify the calculated controller parameters to be the current setting by clicking the **Accept** button.

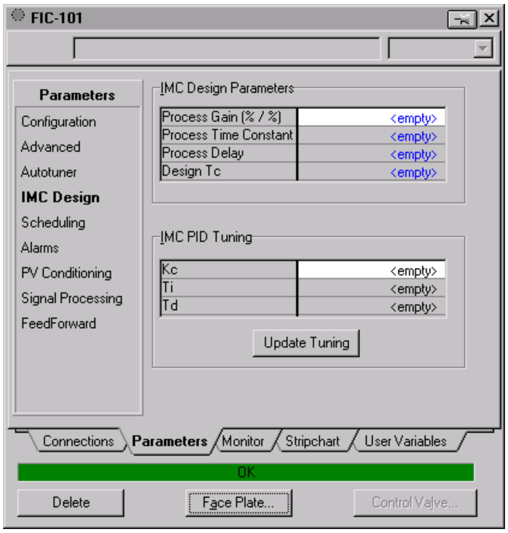
Example

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.

> Authors suggest using the auto tunning 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

Figure 10.11



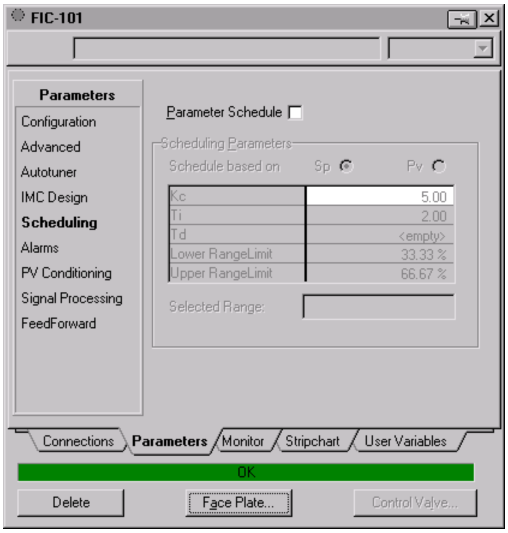
The **IMC Design** page allows the users to use the internal model control (IMC) calculator to calculate the PID parameters based on a supplied model of the process one is attempting to control. The IMC method is quite common in most of the process industries and has a very solid theoretical basis. In 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. The user must supply 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	This group 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	This group 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

Figure 10.12



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. The parameter scheduling is activated through **Parameter Schedule** check box. 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.

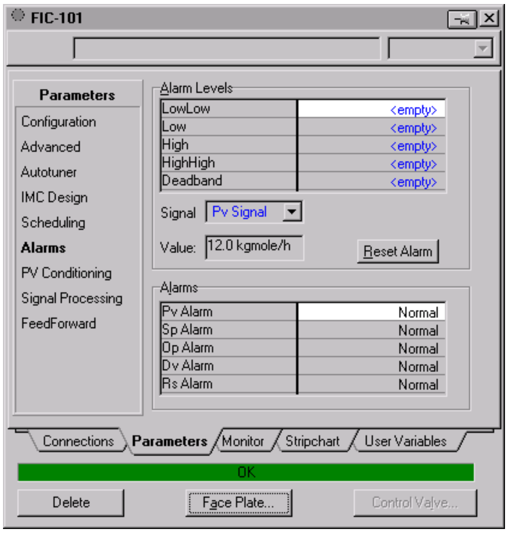
- Low Range
- Middle Range
- High Range

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 Limit**.

> Note that the values of 0 and 100 cannot be specified for both the Lower and the Upper Range Limits

Alarms Page

Figure 10.13



The **Alarms** page allows you to set alarm limits on all exogenous inputs to and outputs from the controller. The page contains two groups and one button:

- Alarm Levels
- Alarms
- Reset Alarm button

Alarm Levels

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.

> Note that the above limits are set internally and are not available for adjustment by the user!

Alarms

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 will be available for the feedforward controller in the Future)
RS	Remote Setpoint

Reset Alarm

When the deadband has been set, it is possible that an alarm status will be triggered and the alarm will not disappear until the band has been exceeded. The **Reset Alarm** button will allow the alarm to be reset when within the deadband.

Example

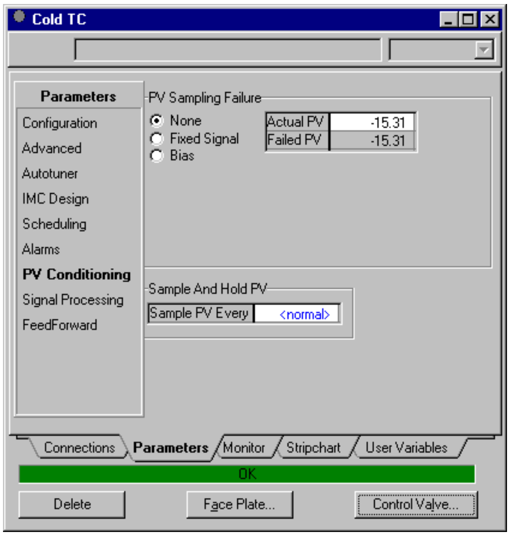
It is desired to control the flowrate through a valve within the operating limits. Multiple alarms can be set to alert you about increases or decreases in the flowrate. For the purpose of this example, you will be specifying low and high alarm limits for the process variable signal. Assuming that the 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 select 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 like 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.

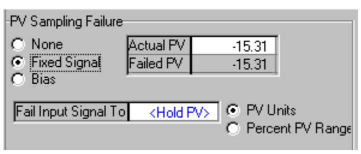
Figure 10.14



This view consists of two groups: **PV Sampling Failure** and **Sample and Hold PV**.

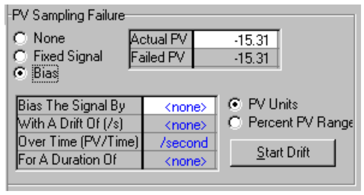
The PV Sampling Failure group consists of three radio buttons: None, Fixed Signal and Bias. The options presented to you changes with respect to the radio button chosen. When the **None** radio button is selected the dialogue is as seen in **Figure 10.14**, with only the Actual PV and Failed PV values displayed. When **Fixed Signal** radio button is selected the PV Sampling Failure group appears as follows.

Figure 10.15



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 10.16



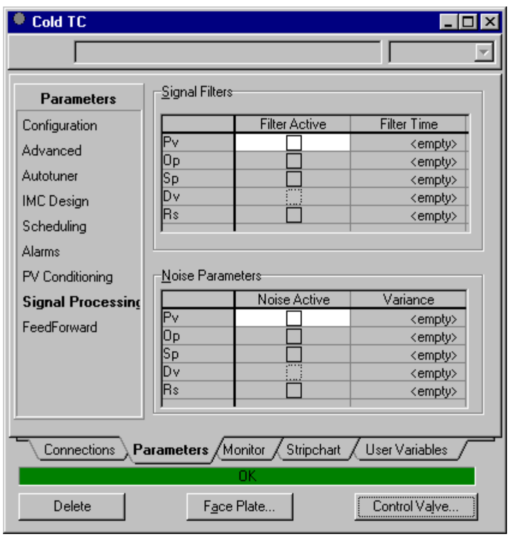
The PV Sampling Failure group now allows you to drift the input signal. The parameters allow you to bias the signal and create a drift over a period of time. To start the drift simply click the **Start Drift** button.

The Sample And Hold PV group allows you to take a PV sample and hold this value for a specified amount of time.

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 10.17



This page of 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

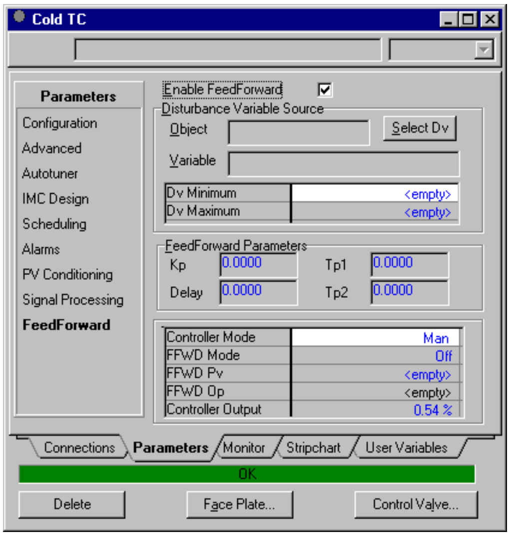
To apply the filter activate the check-box corresponding to the signal you wish to filter. Once active you can specify the filter time. As you increase the filter time you will be filtering out frequency information from the signal. For example is the signal is noisy there will be a smoothing effect noticed on the plot of the Pv. Note that it is possible to add a filter that will make the controller **unstable**. In such cases the controller will need 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. Note 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 will 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 10.18



To enable feedforward control you must activate the **Enable FeedForward** check-box.

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 will open the Variable Navigator, see figure Figure 10.2.

The allows you to specify controller parameters. This group

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 model for the FeedForward controller is as follows:

$$G(s) = K_p \frac{(\tau_1 s + 1)e^{-ds}}{\tau_2 s + 1} \quad (10.8)$$

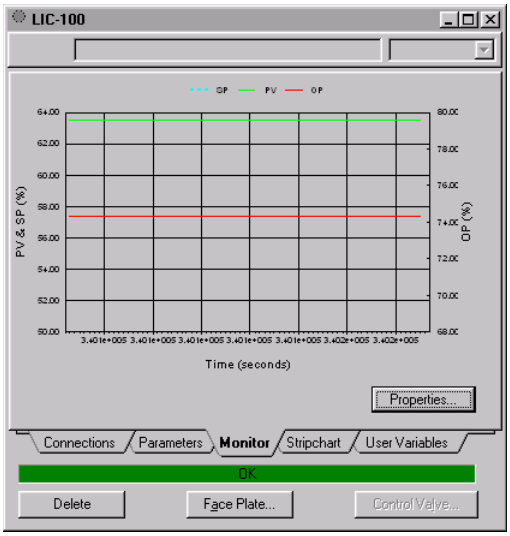
where: K_p = *gain*

τ = *time constant*

d = *deadtime or delay*

10.1.4 Monitor Tab

Figure 10.19



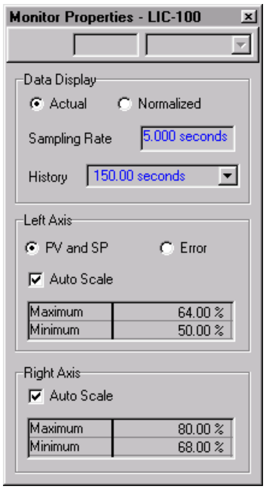
A quick monitoring of the response of the Process Variables, Setpoint and Output can be seen on the **Monitor** tab. This tab allows you to monitor the behaviour of process variables in a graphical format while calculations are proceeding.

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 object inspecting any spot on the plot area. The object inspection menu contains the following options:

Item	Description
Graph Control	Opens the Graph Control view to modify many of the plot characteristics. See **Section 6.4 - Graph Control** of the **User Guide**.
Turn Off/On Cross Hair	Select this item to turn the cross hair either on or on.
Turn Off/On Vertical Cross Hair	Select this item to turn the vertical cross hair either on or on.
Turn Off/On Horizontal Cross Hair	Select this item to turn the horizontal cross hair either on or on.
Values Off/On	Display the current values for each of the variables, if turned on.
Copy to Clipboard	Copy the current plot to the clipboard with the chosen scale size.
Print Plot	Print the plot as it appears on the screen.
Print Setup	Access the typical Windows Print Setup, which 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** view, which can be access by pressing the **Properties** button.

Figure 10.20

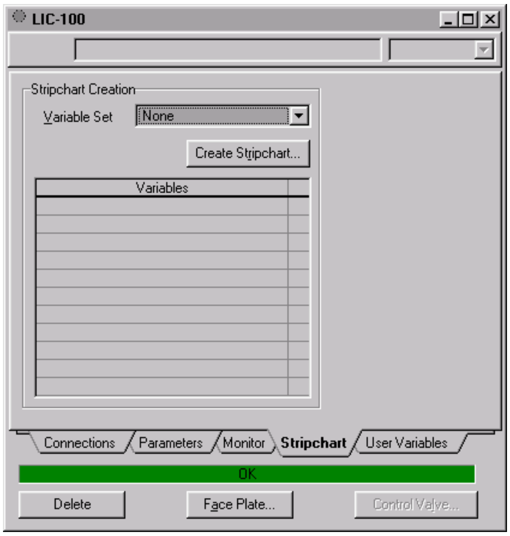


There are three group available on this view and are describe as follows:

Group	Description
Data Capacity	This group allows you specify the type and amount of data to be displayed. You can also select the data sampling rate.
Left Axis	This group give 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	This group give you an option to either customize the right axis scale or let HYSYS auto scale it according to the current OP value.

10.1.5 Stripchart Tab

Figure 10.21



The **Stripchart** tab allows you to setup stripcharts for various variable sets. You can select from the given choices in the **Variable Set** drop down box and then press the **Create Stripchart** button to display the stripchart.

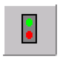
10.1.6 User Variables Tab

The **User Variables** tab allows you to create and implement variables in the HYSYS simulation case. For more information on implementing User Variables, see Chapter 5 - User Variables in the **Customization Guide**.

10.2 Digital Point

The PV is optional; if you do not attach a Process Variable Source, the Digital Point will operate in Manual mode.

The **Digital Point** is an On/Off Controller. You specify the Process Variable (PV) you wish to monitor, and the stream which you are controlling (OP). When the PV reaches a specified threshold value, the Digital Point will either turn the OP On or Off, depending on how you have set up the Digital Point.



Digital Control Point Button

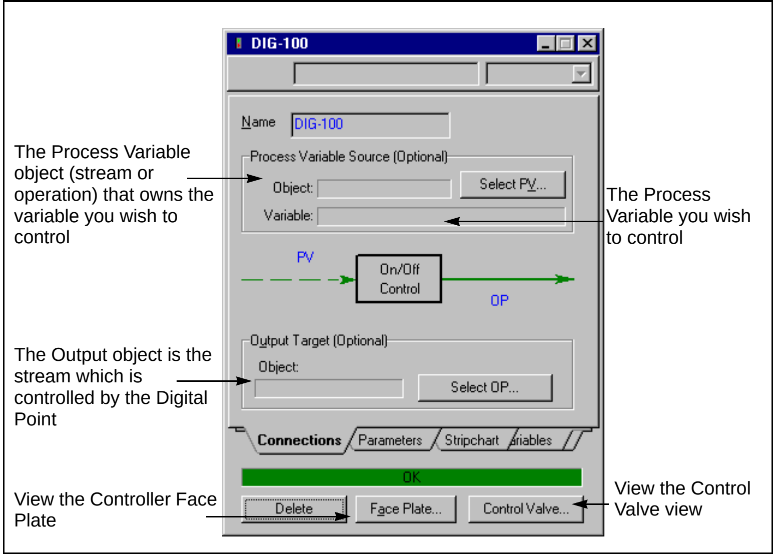
To install the **Digital Point** controller, select **Add Operation** from the **Flowsheet** menu, and select **Digital Control Point**. Alternatively, select the **Digital Control Point** button in the Palette.

10.2.1 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. This controller 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. Note that this configuration can only be used for Manual mode. To run the controller in Automatic mode you requires a Process Variable Source input. With only the input connected the digital point will act 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.

To specify the controller input use the **Select PV** button to access the **Variable Navigator**, which allows you to simultaneously target the PV Object and Variable. Similarly, use the **Select OP** button to choose the **Output Target**.

Figure 10.22



Note that the flow of the OP Output is manipulated by the Digital Point Controller.

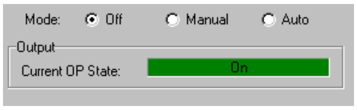
10.2.2 Parameters Tab

The **Parameters** tab provides three different modes of operation: Off, Manual and Auto. For each of these modes the Parameters tab is made up of a number of groups: Output, Operational Parameters and Faceplate PV Configuration**.**

Off Mode

When **Off** mode is selected you will not be able to adjust the OP State. Note that if you turn the controller Off while running the simulation, it will retain 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 this mode.

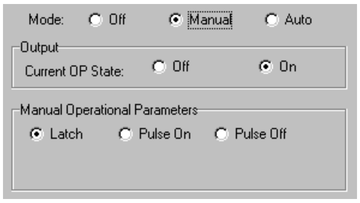
Figure 10.23



Manual Mode

When **Manual** mode is selected you will be able to adjust the OP State from the Faceplate or this tab. Two groups are visible when in this mode: Output and Manual Operation Parameters.

Figure 10.24



The Output group allows you to toggle the OP state on and off. The Operational Parameters group allows you to select one of three options: Latch, Pulse On and Pulse Off.

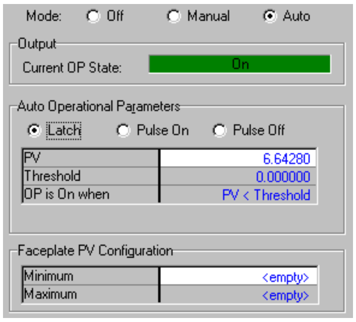
Allows you to specify the duration of the pulse.

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.

Auto Mode

When **Auto** mode is selected HYSYS will automatically operate 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 10.25



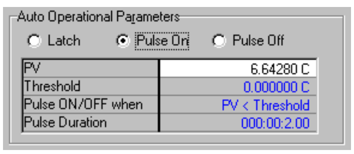
Since HYSYS is automatically adjusting the controller the Output group simply displays the OP State. Like Manual mode the Auto Operation Parameters group allows you to select one of three options: Latch, Pulse On and Pulse Off.

When Latch is selected the following parameters are displayed.

Parameter	Description
PV	This is the actual value of the PV (Process Variable).
Threshold	This is the value of the PV which determines when the controller switches the OP On or Off.
OP is On when	This allows you to set the condition when the OP state is On. This is defined in one of two ways: when the PV is greater than the Threshold, or when the PV is less than the Threshold.

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 10.26



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.

10.2.3 Stripchart Tab

The **Stripchart** tab allows you to setup stripcharts for various variable sets. You can select from the given choices in the **Variable Set** drop down box and then press the **Create Stripchart** button to display the stripchart.

10.2.4 User Variables Tab

The **User Variables** tab allows you to create and implement variables in the HYSYS simulation case. For more information on implementing User Variables, see **Chapter 5 - User Variables** in the **Customization Guide**.

10.2.5 Alarm Levels Tab

The **Alarms** tab allows you to set alarm limits for the controller. 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.

> Note that the above limits are set internally and are not available for adjustment by the user!

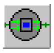
The Alarm Status displays the recently violated alarm for each alarm point.

10.3 MPC

The current version of the MPC implementation does not handle the problem of processes with constraints - a future release will be capable of handling that class of problems.

The “Model Predictive Control“ (MPC) controller address the problem of controlling processes that are inherently multi-variable and interacting in nature, i.e., one or more inputs affects more than one output.

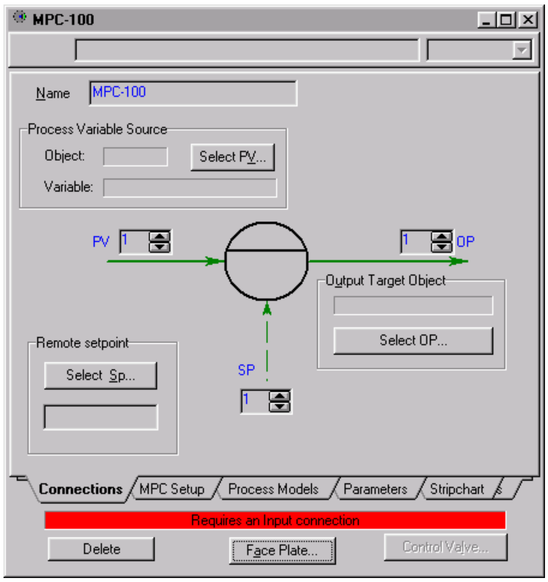
To install **MPC Controller** operation, choose **Add operation** from the **Flowsheet** menu, and select **MPC Controller**. Alternatively, select the **MPC Controller** button from the Palette.



MPC Controller Button

10.3.1 Connections Tab

Figure 10.27



The **Connections** tab is comprised of six objects that allow you to select the Process Variable Source, Output Target Object and Remote SP.

Object	Description
Name	This field contains the name of the controller. It can be edited by selecting the field and entering the new name in the Edit Bar.
Process Variable Source Object	This field contains the Process Variable Object (stream or operation) that owns the variable you wish to control. It is specified via the Variable Navigator.
Process Variable	This field contains the Process Variable you wish to control.
Output Target Object	The Output Target Object field is 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 allows you to select a specific Process Variable, Output Target Object, and Remote Setpoint Source respectively.

Process Variable Source

The Process Variable, or PV, is the variable that must be maintained or controlled at a desired value.

Common examples of PVs include vessel pressure and liquid level, as well as stream conditions such as flow rate or temperature.

The Variable Navigator allows you to simultaneously select the Object and Variable.

To attach the **Process Variable Source**, click the **Select PV** button. Then select the appropriate object and variable simultaneously, using the Variable Navigator (**Figure 10.2**). For more information, see **Section 5.2 - Navigation** of the User's Guide.

Remote Setpoint

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, i.e., a master in the classical cascade control scheme.

The **Select Sp** button allows you to select the remote source using the Variable Navigator.

Output Target Object

> The Output of the Controller is the control valve that 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** view.

The Controller compares the Process Variable to the Setpoint and produces an output signal which causes the control valve to open or close appropriately.

Selecting the **Output Target Object** is done in a similar manner to selecting the **Process Variable Source**. In this case, however, you are only selecting the Object. Only Objects with Control Valves not currently attached to another Controller will be accessible.

The information regarding the Valve sizing is contained on a Sub-View accessed via the **Control Valve** button found at the bottom of the **MPC Controller** property view.

10.3.2 Control Valve

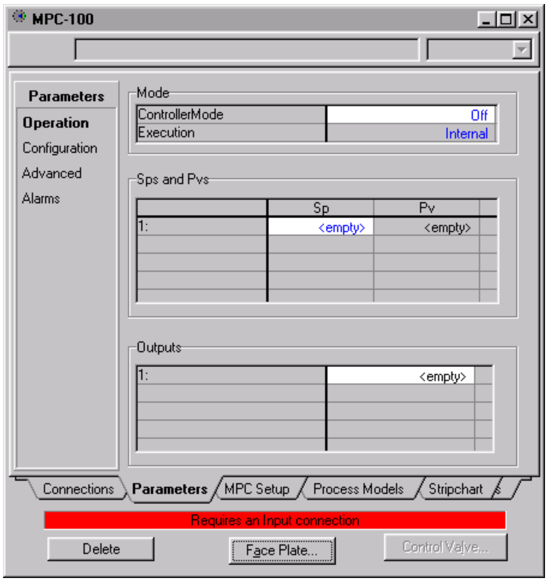
The information shown on the **Control Valve** view is specific to the associated valve. Refer to section **Section 10.1.2 - Control Valve** for more information.

10.3.3 Parameters Tab

The Parameters tab consist of four pages: **Operation**, **Configuration**, **Advanced** and **Alarms**.

Operation Page

Figure 10.28



The Operation page allows you to set the Execution Type, Controller Mode and depending on the mode, either the SP or OP.

Mode

The Controller will operate 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	Manipulate 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.

The mode of the controller may also be set on the Face Plate, refer to Section 10.7 - Controller Face Plate for more information.

Sps and Pvs

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.

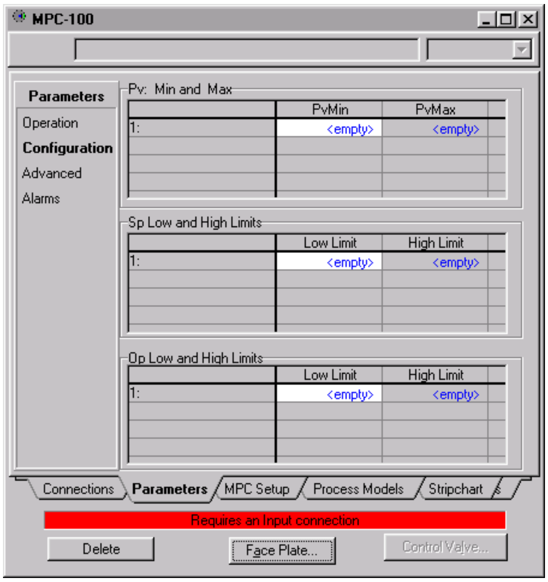
Outputs

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 input a value for the Output, and the Setpoint will become 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

Figure 10.29



The **Configuration** page allows to specify the Process Variable, Setpoint and Output ranges.

PV: Min and Max

For the Controller to become operational, you must:

Note that without a PV span, the Controller will not function.

1. Define the minimum and maximum values for the PV (the Controller will not 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. Note that HYSYS will use the current value of the PV as the set point by default, but you may change this value at any time.
3. 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:

$$PV(\%) = \left(\frac{PV - PV_{min}}{PV_{max} - PV_{min}}\right)100 \qquad (10.9)$$

SP Low and High Limits

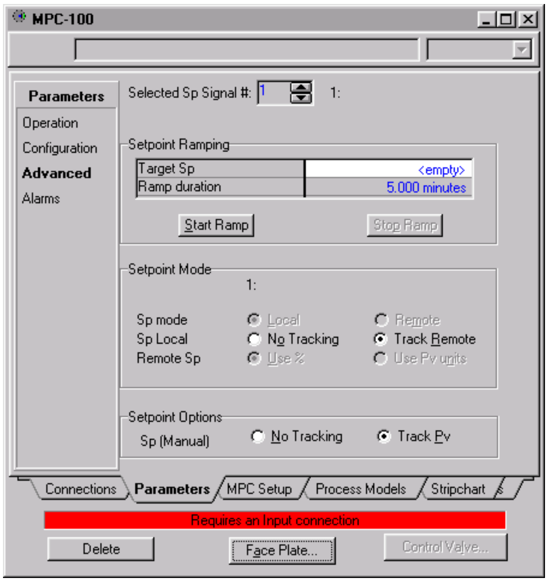
Specify the higher and lower limits for the Setpoints to reflect your needs and safety requirements. The Setpoint limits will enforce an acceptable range of values that could be entered via the interface or from a remote source. By default the PVs min. and max values will be used as the SPs low and high limits, respectively.

Op Low and High Limits

Specify the higher and lower limits for all the outputs. The output limits will 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.

Advanced Page

Figure 10.30



The **Advanced** page contains the following three groups:

Group	Description
Setpoint Ramping	This group allow you to specify the ramp target and duration.
Setpoint Mode	This group contains the options for setpoint mode and tracking as well as the option for remote setpoint.
Setpoint Options	This group contains the option for setpoint tracking only in manual mode.

The setpoint signal is specified in the **Selected Sp Signal #** box by:

- clicking the up or down arrow button
- or typing the appropriate number in the field.

Depending upon the signal selected, the page will display the respective setpoint settings.

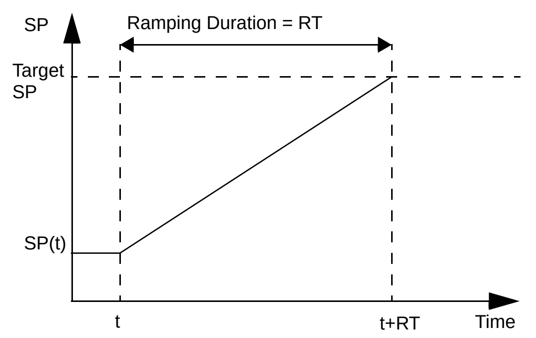
Setpoint Ramping

Setpoint ramping is only available in Auto mode.

The setpoint ramping function has been modified in the present MPC controllers. Now it is continuous, i.e., when set to on by clicking the **Start Ramp** button, the setpoint will change over the specified period of time in a linear manner. The **Setpoint Ramping** group contains the following two fields:

Field	Input Required
Target SP	This field contains the Setpoint you wish 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	This field contains the time interval you wish to complete setpoint change in.

Figure 10.31



Controller Ramping

Besides these two fields there are also two buttons available in this group:

Button	Functionality
Start Ramp	Pressing this button activates the ramping process.
Stop Ramp	Pressing this button 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 will show the final value of the setpoint whereas the **SP** field, on the **Operation** page, will show the current setpoint seen internally by the control algorithm. Note that during ramping, if a second setpoint change has been activated, then **Ramping Duration** time would be restarted for the new setpoint.

Example

If you click the **Start Ramp** button and enter values for the two parameters in the **Setpoint Ramping** group, the Controller will switch to Ramping mode and adjust the Setpoint linearly (to the **Target SP**) during the Ramp Duration, see **Figure 10.31**. For example, suppose your current SP is 100 and you wish to change it to 150. Rather than creating a sudden large disruption by manually changing the SP while in Automatic mode, click the **Start Ramp** button and enter the 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 will adjust the SP from 100 to 150 linearly over the 10 minute interval.

Setpoint Mode

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 user can manually specify the setpoint via the 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, i.e., 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 activating the **Track Remote** button.

The **Remote Sp** option allows activate 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 units

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

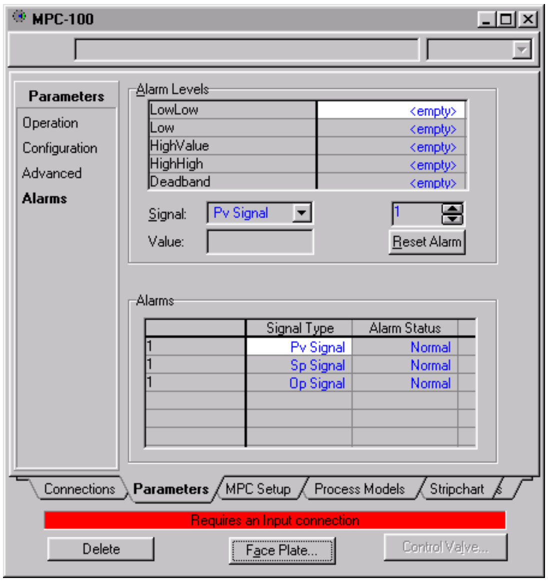
If the **Track PV** radio button is selected then there is automatic setpoint tracking in manual mode, that sets the value of the setpoint equal to the value of the Pv prior to the controller being placed in the manual mode. This means that upon switching from manual to automatic mode the values of the setpoint and Pv were equal and, therefore, there was an automatic bumpless transfer. Also you have the option not to track the pv, by activating 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 will be an instantaneous bumpless transfer prior to the controller recognizing a setpoint that is different from the Pv.

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 will internally convert the new setpoint as m^3/s i.e. 1/3600 = 0.00028 m^3/s and pass it to the controller, which will read the value and convert it back into m^3/h i.e. 1 m^3/h. The controller will use this value as the new setpoint. If the units are not specified, then the spreadsheet will pass it as 1 m^3/s, which is the base unit in HYSYS, and the controller will convert 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 will use the maximum value i.e. 1.75m^3/h as the new setpoint.

Alarms Page

Figure 10.32



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

Alarm Levels

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, 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, the user has 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.

> Note that the above limits are set internally and are not available for adjustment by the user!

Alarms

The **Alarms** group display the recently violated alarm for the following signals:

Signal	Description
PV	Process Variable
OP	Output
SP	Setpoint

Example

It is desired to control the flowrate through a valve within the operating limits. These limits can be monitor using the Alarms feature in MPC Controller. Multiple alarms can be set to alert the user about increase or decrease in the flowrate. For the purpose of this example, you will be 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.

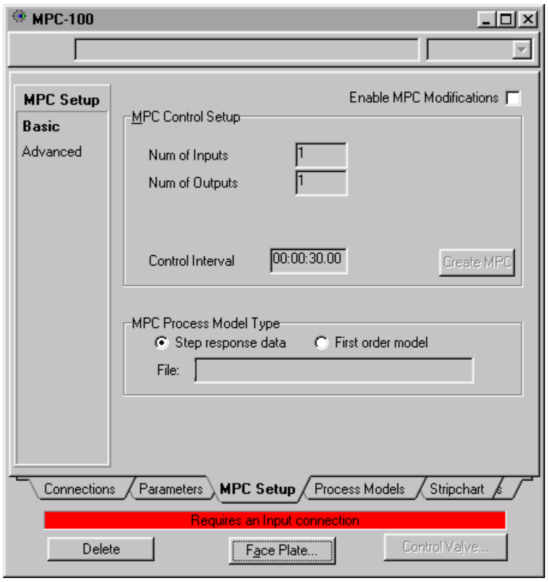
To set the low alarm, first make sure that the **Pv Signal** is selected in the **Signal** drop down list. Specify a value of 0.7 m^3/h in the cell beside the Low alarm level. Follow the same procedure to specify a High alarm limit at 1.5 m^3/h. If you like to re-enter the alarms, click the **Reset Alarm** button to erase all the previously specified alarms.

10.3.4 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 will be an effective and efficient controller.

Basic Page

Figure 10.33



The **Basic** page divides the setup settings into **MPC Control Setup** and **MPC Process Model Type** groups.

MPC Control Setup

In the **MPC Control Setup** group the user is required to supply the following:

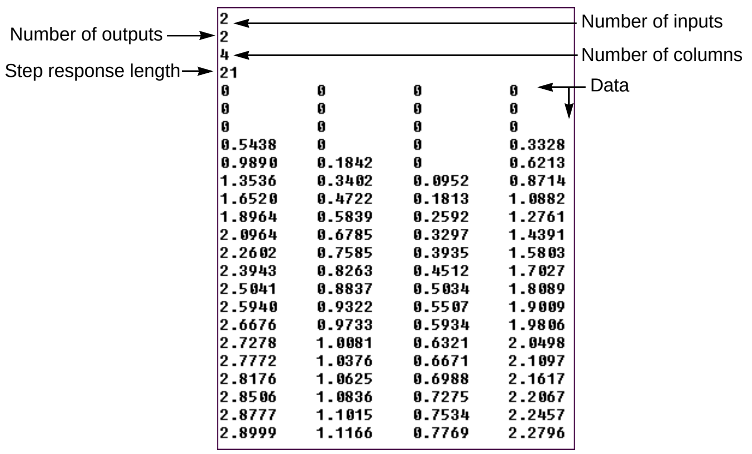
*Note that anytime one of the MPC setting is changed a new MPC object has to be created internally-this is automatically achieved by clicking on the "**Create MPC**" button.*

Field	Description
Num Inputs	Specify the number of process input. Up to a maximum of 12 process inputs can be specified. The default value is 1.
Number Outputs	Specify the number of process output. Up to a maximum of 12 process inputs can be specified. The default value is 1.
Control interval	Specify the control or sampling interval. The default value is 30 seconds.

MPC Process Model Type

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:

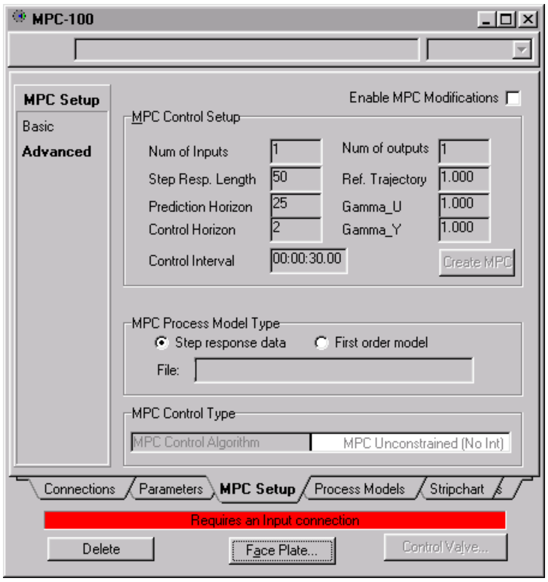
Figure 10.34



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

Figure 10.35



The **Advanced** page divides the setup settings into **MPC Control Setup**, **MPC Process Model Type** and **MPC Control Type** groups.

MPC Control Setup

In the **MPC Control Setup** group the user is required to supply the following:

*Note that 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.*

Field	Description
Num Inputs	Specify the number of process input. Up to a maximum of 12 process inputs can be specified. The default value is 1.
Number Outputs	Specify the number of process output. Up to a maximum of 12 process inputs can be specified. The default value is 1.
Control interval	Specify the control or sampling interval. The default value is 30 seconds.
Step response length	Specify 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	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	Specify the number of control moves into the future that will be 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	Specify the time constant of a first order filter that operates on the true setpoint. A small reference trajectory lets the controller sees a pure step as the setpoint is changed. The default value is 1.
Gamma_U/ Gamma_Y	Specify the positive-definite weighting functions, which are associated with the optimization problem that is solve to produce the controller output every control interval. The value of Gamma_U and Gamma_Y should be between 0 and 1. The default values are 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 will make it's predictions, based on the supplied 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 will look 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 will be. 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 supplied 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:

$$1 \leq \text{Reference Trajectory} \leq 15 \times \text{Sampling Interval}$$

MPC Process Model Type

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 10.34**.

MPC Control Type

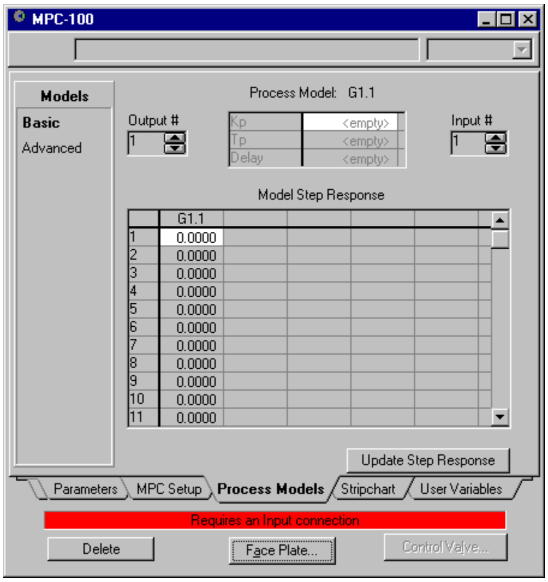
This group allows you to select the **MPC Control Algorithm** that will be used by the controller. At Present, the only option available for selection is the **MPC Unconstrainted (No Int)**. This algorithm does not consider constraints on either controlled and manipulated variables.

10.3.5 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 10.36



Step Response Data

*You cannot modify the model step response data on the **Process Model** tab.*

If the **Step response data** radio button is selected on the **MPC Setup** tab, the **Process Model** tab displays the Model Step Response matrix.

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 is displayed.

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 box by:
   - clicking the up or down arrow button.
   - typing the appropriate number in the field.

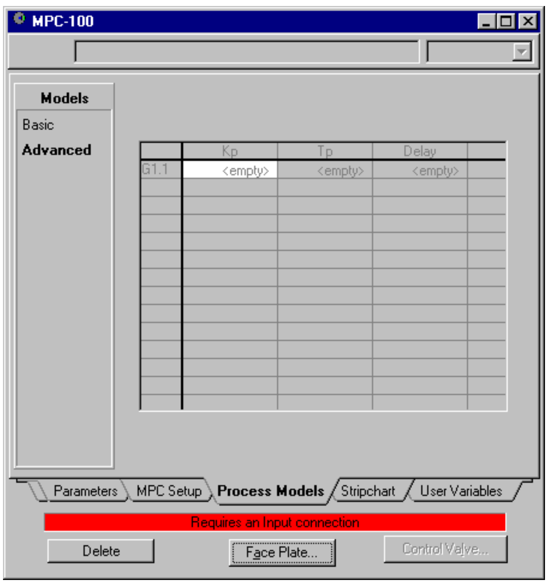
   Depending upon the input and output variable selected, the relevant process model will be displayed.
2. Then specify the process gain (Kp), process time constant (Tp) and delay for the selected process model in the available matrix.
3. Repeat step 1-2 for the remaining process models.

Then click the **Update Step Response** button to calculate the step response data for the process models.

Advanced Page

The Advanced page list all of the Process Models and their associated tuning parameters in table.

Figure 10.37



10.3.6 Stripchart Tab

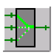
The **Stripchart** tab allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the **Create Stripchart** button to display the stripchart.

10.3.7 User Variables Tab

The **User Variables** tab allows you to create and implement variables in the HYSYS simulation case. For more information on implementing User Variables, see **Chapter 5 - User Variables** in the **Customization Guide**.

10.4 Selector Block

The **Selector Block** is a multiple-input single-output controller, that provides signal conditioning capabilities. It will determine 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.

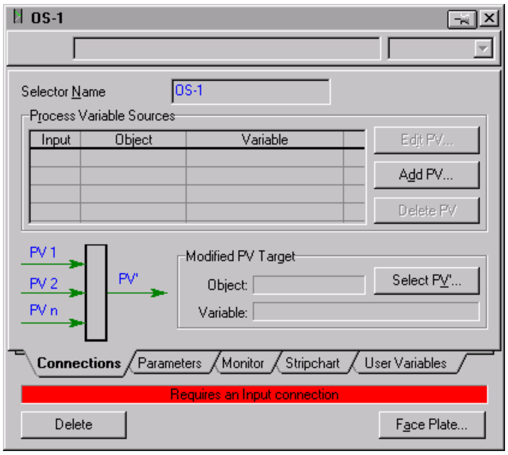


Selector Button

To install the **Selector Block** operation, select **Add Operation** from the Flowsheet Menu, and choose **Selector Block**. Alternatively, choose the **Selector Block** button in the Palette.

10.4.1 Connections Tab

Figure 10.38

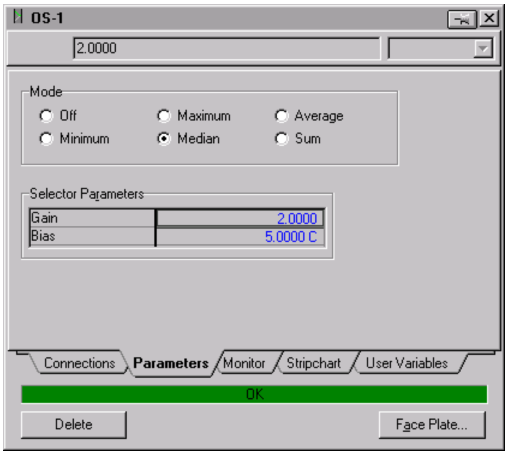


The **Connections** tab consists of three objects:

Objects	Description
Selector Name	This field contains the name of the Selector Block which can be edited at any time.
Process Variable Sources	This group contains the variables the Selector Block will consider and inserted into the input function.The Process Variables for the various inputs are targeted by clicking **Add PV**, accessing the Variable Navigator (see **Section 5.2 - Navigation** of the **User's Guide** for details). 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 will display an error message. However, you will be allowed to retain that Variable.
Modified PV Target	This group contains the process variable which will be manipulated by the Selector Block. To select the **Modified PV Target** click the **Select PV** button. This button also accesses the Variable Navigator. Note that it is not necessary for the Target Variable type to match the Input Variable type.

10.4.2 Parameters Tab

Figure 10.39



The Output is a function of the Mode, Gain and Bias, where the Input Function is dependent on the Mode:

$$Output = f(Inputs) \times Gain + Bias \tag{10.10}$$

The Input Function is multiplied by the Gain. In effect, the gain tells how much the output variable will change per unit change in the input function.

The Bias is added to the product of the Input Function and Gain. If you wish to view the Input function without any Gain or Bias adjustment, set the Gain to one and the Bias to zero.

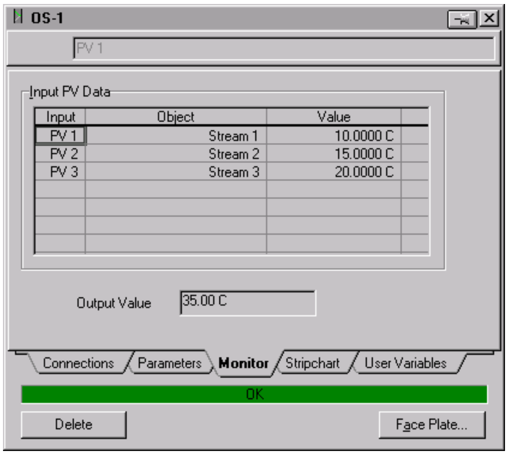
You may choose from the following Modes:

Modes	Description
Minimum	The minimum value from the list of Input Variables will be passed to the Output stream.
Maximum	The maximum value from the list of Input Variables will be passed to the Output stream.
Median	The median value of the Input Variables will be passed to the Output stream. If there are an even number of Input Variables, then the higher of the two middle values will be passed to the Output stream.

Modes	Description
Average	The average of the Input Variables will be passed to the Output stream.
Sum	The sum of the Input Variables will be passed to the Output stream.

10.4.3 Monitor Tab

Figure 10.40



The **Monitor** tab displays the results of the Selector Block. It consists of two objects: the **Input PV Data** group box, that contains the current values of the input process variable and the **Output Value** field, that contains the current value of the PV for each of the input variables and the Output Value is also displayed on the **Parameters** tab.

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:

- 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.
- Determine the Output Value, from the Gain and Bias. The Gain is 2, and the Bias is 5 °C.

The Output is calculated as follows:

$$\begin{aligned} Output &= f(Inputs) \times Gain + Bias \\ Output &= 15°C \times 2.000 + 5.000°C \\ Output &= 35°C \end{aligned} \tag{10.11}$$

10.4.4 Stripchart Tab

The **Stripchart** tab allows the user to setup stripchart for various variables sets. You can select from the given choices in the **Variable Set** drop down box and then press the **Create Stripchart** button to display the stripchart.

10.4.5 User Variables Tab

The **User Variables** tab allows you to create and implement variables in the HYSYS simulation case. For more information on implementing User Variables, see **Chapter 5 - User Variables** in the **Customization Guide**.

10.5 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 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:

$$Y = MX + B \tag{10.12}$$

where: Y = *Dependent (target) variable*

X = *Independent (source) variable*

M = *Multiplier (slope)*

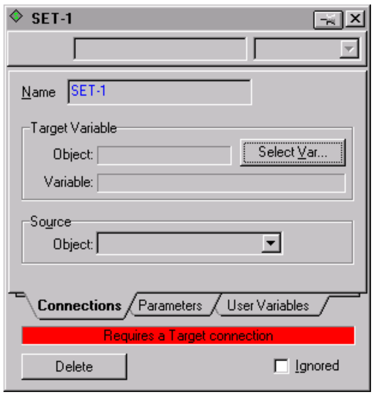
B = *Offset (intercept)*

Set Button

To install the **Set** operation, choose **Add Operation** from the **Flowsheet** menu, and select **Set**. Alternatively, select the **Set** button in the Object Palette.

10.5.1 Connections Tab

Figure 10.41



On the **Connections** tab, specify the following information:

Input Required	Description
Target Object	The stream or operation that the dependent variable belongs. This is specified by clicking the **Select Var** button. This will bring up the Variable Navigator (see **Section 5.2.2 - Variable Navigator** of the **User Guide** for more information about using the Variable Navigator)

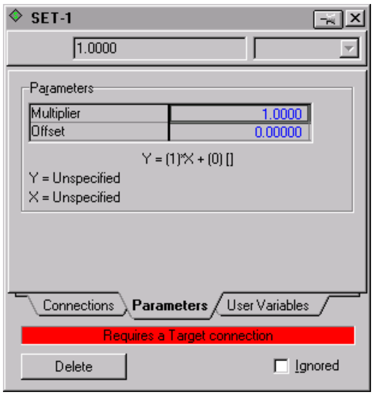
Input Required	Description
Target Variable	The type of variable you wish to set, i.e., temperature, pressure, flow. The available choices for *Variable* are dependent on the Object type (stream, heat exchanger, etc.) Your choice of *Variable* will automatically be assigned to both the Target and Source object.
Source Object	The stream or operation to which the independent variable belongs.

Note 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 will be available for the Source.

HYSYS will solve for either the Source or Target variable, depending on which is known first (bi-directional solution capabilities).

10.5.2 Parameters Tab

Figure 10.42



The **Parameters** tab allows you to provide values for the **Slope** (Multiplier, or ’M’-value) and the **Intercept** (Offset, or ’B’-value in the set equation). The default values for the Multiplier and Offset are 1 and 0, respectively.

To ignore the **Set** during calculations, select the **Ignore** checkbox. HYSYS will completely disregard the operation until you restore it to an active state by clearing the check box.

10.5.3 User Variables Tab

The **User Variables** tab allows you to create and implement variables in the HYSYS simulation case. For more information on implementing User Variables, see **Chapter 5 - User Variables** in the **Customization Guide**.

10.6 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
- Ramp

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:

$$G(s) = Lead1(s)Lead2(s)Lag1(s)Lag2(s)W(s)D(s)R(s) \tag{10.13}$$

The input X(s) is multiplied by the transfer function to obtain the output. Note that the input (or Process Variable Source) is optional; you can use a fixed value as the input.

$$Y(s) = G(s)X(s) \tag{10.14}$$

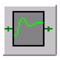
The transfer function is defined here in the Laplace Domain (using the Laplace Variables). When in the Laplace Domain, the overall transfer function is simply the product of the individual transfer functions.

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:

$$G = \frac{\omega}{s^2 + \omega^2} \tag{10.15}$$

When converted to the time domain by taking the inverse Laplace, we obtain:

$$f(t) = \sin \omega t \tag{10.16}$$

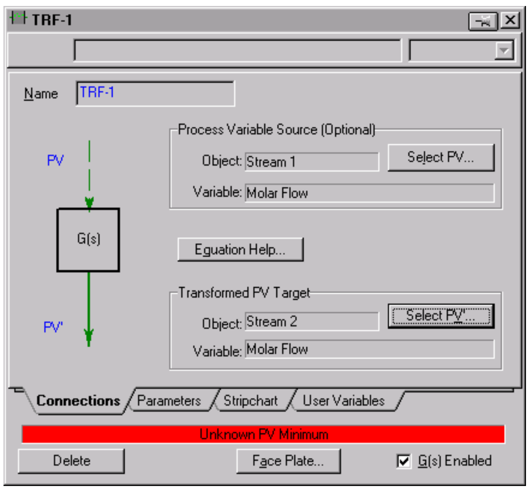


Transfer Function Button

To install the **Transfer Function** operation, choose **Add Operation** from the **Flowsheet** menu, and select **Transfer Function Block**. Alternatively, select the **Transfer Function Block** button in the Object Palette.

10.6.1 Connections Tab

Figure 10.43



The following information is shown on the **Connections** tab:

Input Required	Description
Name	The name of this Transfer Function Block.
Process Variable Source	This is a stream or Operation. You may select the PV Object and Variable by clicking the **Select PV** button (see **Section 5.2 - Navigation** of the User's Guide). Note that the Process Variable is optional. If you do not specify a PV, enter a constant PV on the **Parameters** tab.

Input Required	Description
Transformed PV Target Object	This is 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.

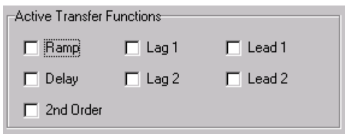
Select the **Eqn Help** button to view the Transfer Function equations.

10.6.2 Parameters Tab

The **Parameters** tab allows you to define the entire transfer function G(s), by defining the Ramp, Delay, Lag, Lead, and 2nd Order transfer functions. The tab consists of the **Configuration** page in which the Process and Output Variable limits are defined.

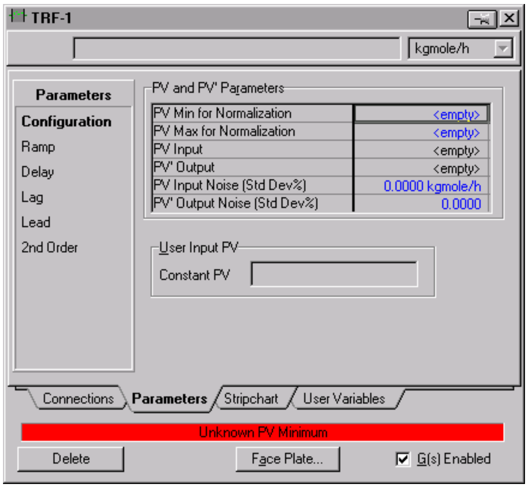
The remaining pages: **Ramp**, **Delay**, **Lag**, **Lead** and **2nd Order** all define the different transfer function terms. Each of these pages contains the **Active Transfer Function** group which consists of a number of check boxes corresponding to the available components of the Transfer Function. By checking the appropriate box you may include that term in the overall Transfer Function. When you activate individual functions on the **Ramp/Delay/Lag or Lead/2nd Order** pages, the appropriate boxes will then be checked in this group.

Figure 10.44



Configuration Page

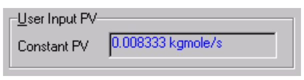
Figure 10.45



Parameter	Description
PV Minimum and Maximum for Normalization	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 will always stay in this range. This range affects the Noise and sine wave amplitude.
PV Input	The value of the PV Input (Process Variable or Constant PV) is shown here.
PV' Output	The value of the PV Output is shown here.
PV Input Noise	Enter the Standard Deviation of the input noise as a percentage of the PV Range.
PV Output Noise	Enter the Standard Deviation of the output noise as a percentage of the PV Range. Note that the noise follows a normal distribution.

User Input PV

Figure 10.46



This box is only visible when you have not defined a PV Source with the **Transformed PV Target** fully defined on the **Connections** tab. You may supply a Constant PV when this box is visible.

Transfer Functions

Create the stream defined in the following table. This stream will be used is the next sections to demonstrate the different transfer functions. The Ramp and Delay Transfer Functions will be demonstrated using a constant value input, while the rest of the Transfer Functions (Lead, Lag, 2nd order Lag, Sine Wave) will be demonstrated using the stream Input as the PV Source.

1. Create the following Fluid Package.

Property Package	Components
UNIQUAC	Methanol and H2O

2. Create the stream Input with the following properties.

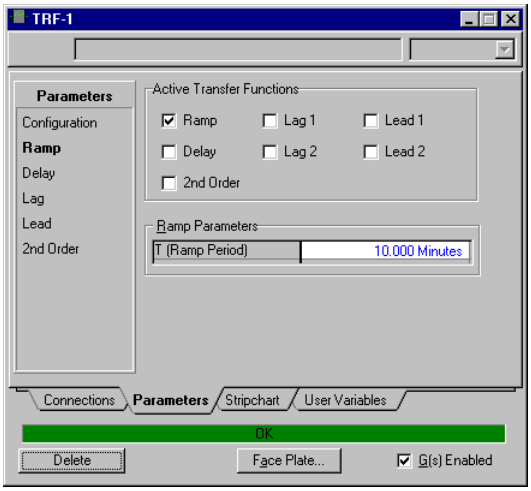
Tab [Page]	Input Area	Entry
Worksheet [Conditions]	Stream Name	Input
	Temperature [C]	25.0000
	Pressure [kPa]	101.3250
	Molar Flow [kgmole/hr]	100.0000
Worksheet [Composition]	Comp Mass Frac [H2O]	0.8000
	Comp Mass Frac [Methanol]	0.2000
Dynamic [Specs]	Pressure Specification	Active
	Flow Specification	Not Active

3. Switch to Dynamics mode.

Ramp Page

The **Ramp** page consists of two group boxes: **Active Transfer Function** and **Ramp Parameters**, that consists of the **T (Ramp Period)** field.

Figure 10.47



The Ramp Transfer Function requires only one parameter T (Ramp Period):

$$G = \frac{1}{Ts} \tag{10.17}$$

The unit step response of the Ramp Transfer Function is

$$f(t) = \frac{t}{T} \tag{10.18}$$

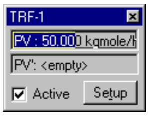
1. Install a Transfer Function Block with the following connections:

Tab [Page]	Input Area	Entry
Connections	Transformed PV Target Object	Input
	Transformed PV Target Variable	Molar Flow

2. On the **Ramp** page of the **Parameters** tab activate the Ramp check-box and specify a value of 10 minutes for the Ramp Period.
3. On the **Configuration** page of the **Parameters** tab enter a PV Range of 0 to 100 kgmole/h.

4. Input a **Constant PV** value of 50. Click the **Face Plate** button at the bottom of the Transfer Function property view

Figure 10.48

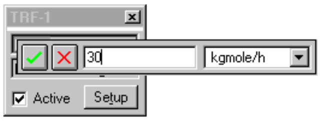


5. Activate the Ramp Transfer Function

Integrator Button

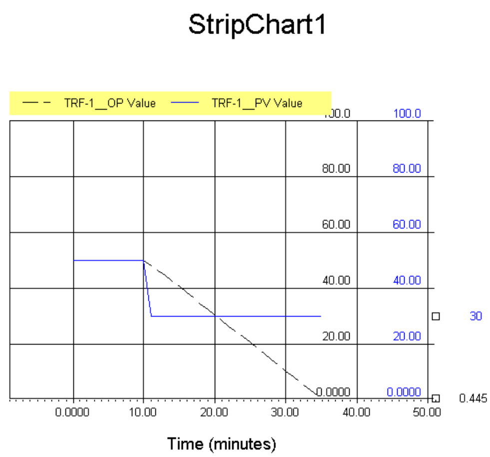
6. Start the Integrator. Initially, the Input and Output remain constant at 50.
7. Change the **Constant PV** value via the Face Plate to 30.

Figure 10.49



8. The Output ramps from a value of 50 to 30 in 10 minutes, and continues to ramp until it reaches the PV Minimum. Note that the Ramp Function is linear, as evidenced in both the graph and the real time Ramp Function.

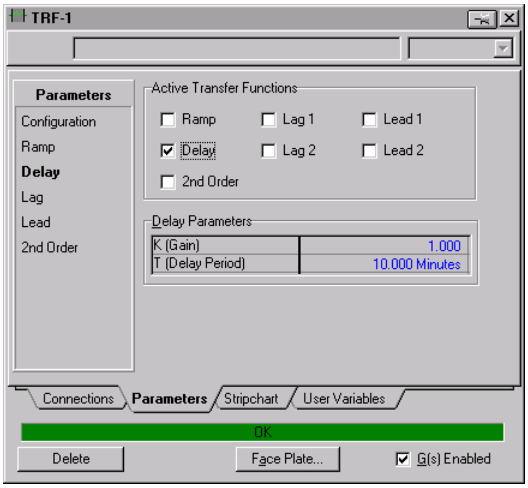
Figure 10.50



Delay Page

The **Delay** Page consists of the **Active Transfer Function** and **Delay Parameters** groups.

Figure 10.51



The **Delay Parameters** group contains two input fields: **K (Gain)** and **T (Delay Period)** that are two parameters of the Delay Equation. The Delay Equation is defined as:

$$G = Ke^{-t_o s} \tag{10.19}$$

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:

$$L^{-1}(Ke^{-t_o s}F(s)) = Kf(t-t_o) \tag{10.20}$$

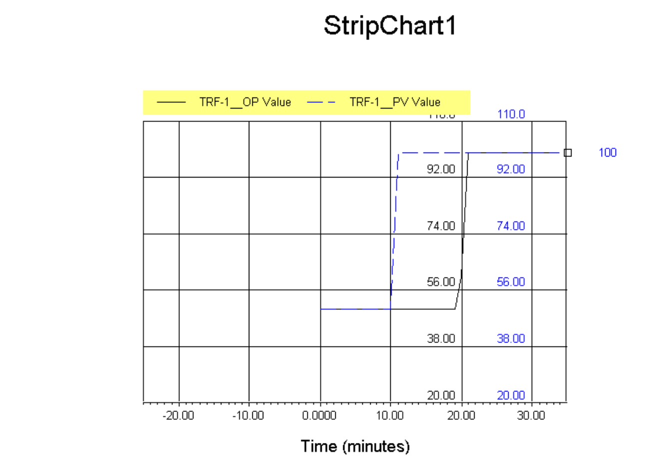
Delay can be used in combination with the other Transfer function terms, by checking the **Delay** check box in the **Active Transfer Function** group. For this example make sure that the **Delay** check box is the only active option.

1. Install a Transfer Function Block with the following connections:

Tab [Page]	Input Area	Entry
Connections	Transformed PV Target Object	Input
	Transformed PV Target Variable	Molar Flow

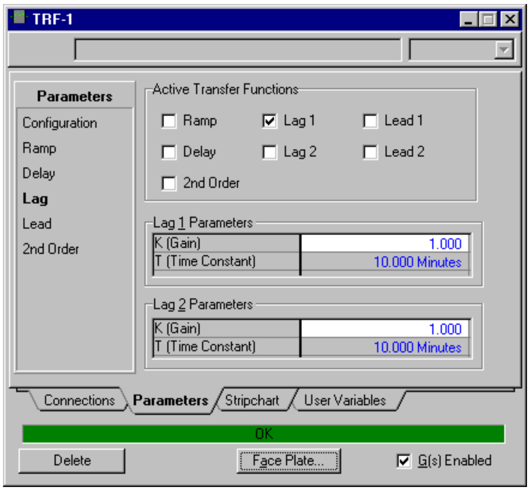
2. On the **Delay** page of the **Parameters** tab activate the Delay check-box and enter the following parameters: **Gain** 1 and **Delay Period** 10 minutes.
3. On the **Configuration** page of the **Parameters** tab enter a PV Range of 0 to 100 kgmole/h.
4. Input a **Constant PV** value of 50. Click the **Face Plate** button at the bottom of the Transfer Function property view
5. Start the Integrator.
6. At a time of about 10 minutes, increase the PV Input from 50 to 100 from the **Face Plate**.
7. At time = 20 minutes, ten minutes after the Input changed, the Output instantaneously increases to 100. The graph shown here displays the Input (dashed line) and Output (solid line) functions.

Figure 10.52



Lag Page

Figure 10.53



You may simulate the response of a first-order or second-order lag on the **Lag** page of the **Parameters** tab. A second order lag can be defined on the **Lag** page creating two first-order lags.

The **Lag** page contains two groups: **Lag 1 Parameters** and **Lag 2 Parameters** with each box defining a single-order lag transfer function. The group contains two fields: **K (Gain)** and **T (Time Constant)**.

The Lag Equation is defined as follows:

$$G = \frac{K}{Ts + 1} \qquad (10.21)$$

where: G = The 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:

$$K\left(1 - e^{-\frac{t}{T}}\right) \tag{10.22}$$

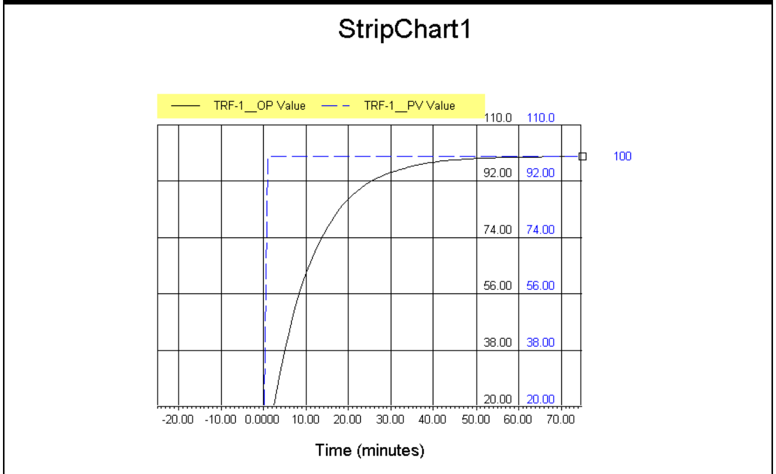
Note that a second-order Lag may also be defined on the **2nd Order** page. This second-order Lag is defined using variables K, T, and ξ (see **Second Order Lag** for details).

1. Define the stream **Input** as shown earlier in this section, and set its flow to zero.
2. Install a Transfer Function Block with the following connections:

Tab [Page]	Input Area	Entry
Connections	Transformed PV Target Object	Input
	Transformed PV Target Variable	Molar Flow

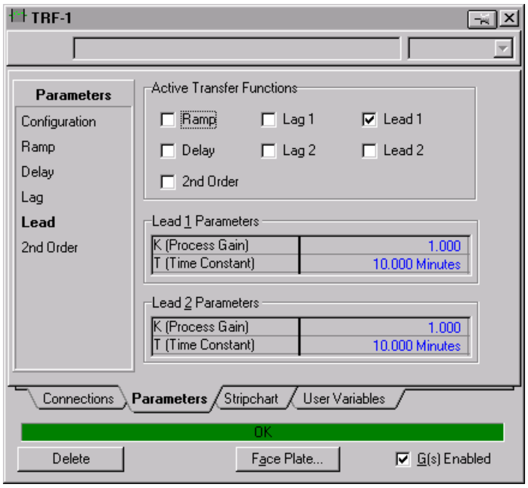
3. On the **Lag** page of the **Parameters** tab activate the Lag 1 check-box and enter the following parameters in the Lag 1 Parameters group: **Gain** 1 and **Time Constant** 10 minutes.
4. On the **Configuration** page of the **Parameters** tab enter a PV Range of 0 to 100 kgmole/h.
5. Start the Integrator. Change the molar flow rate of **Input** to 100 kgmole/h.
6. The first-order response is shown here.

Figure 10.54



Lead Page

Figure 10.55



The **Lead** page allows you to define either a first or second order Lead transfer function. This is done via the two groups: **Lead 1 Parameters** and **Lead 2 Parameters**. Both groups allows for the definition of the two terms of the Lead Equation K and T.

The Lead Equation is defined as:

$$G = K(Ts + 1) \tag{10.23}$$

where: K is the gain and T is the time constant.

The Inverse Laplace Transform of the Lead Equation multiplied by the general function F(s) is:

$$L^{-1}[K(Ts + 1)F(s)] = K\left[\frac{df(t)}{dt}T + f(t)\right] \tag{10.24}$$

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 checking the **Lead** box on the **Parameters** tab.

The response is an exponential curve of the following form:

$$Flow(t) = K\left(1 - e^{-\frac{t}{T}}\right) \tag{10.25}$$

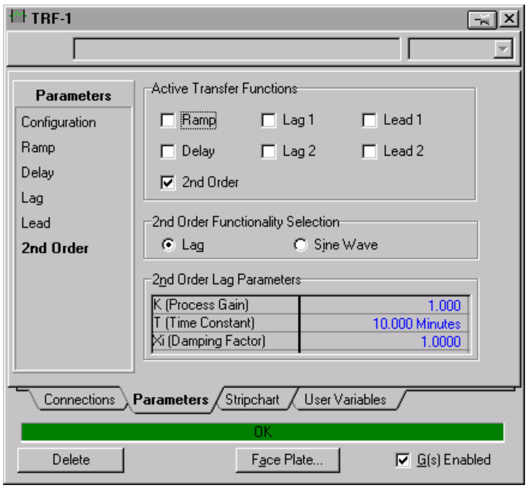
where: K = Process gain

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.

Second Order Lag

Figure 10.56



Standard Second Order

Select the **Lag** radio button in the **2nd Order Functionality Selection** group is used to simulate the response of a standard Second Order process.

The Second Order Lag is defined as:

$$G = \frac{K}{T^2 s^2 + 2T\xi s + 1} \tag{10.26}$$

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 will overshoot the target value, and will oscillate with a period T.

Check the **2nd Order** check box to simulate the Second Order Process. Note that First Order, Delay and Ramp Functions may also be active, in which case all equations will be superimposed.

1. Define the stream **Input** as shown earlier in this section, and set its flow to zero.
2. Install a Transfer Function Block with the following connections:

Tab [Page]	Input Area	Entry
Connections	Transformed PV Target Object	Input
	Transformed PV Target Variable	Molar Flow

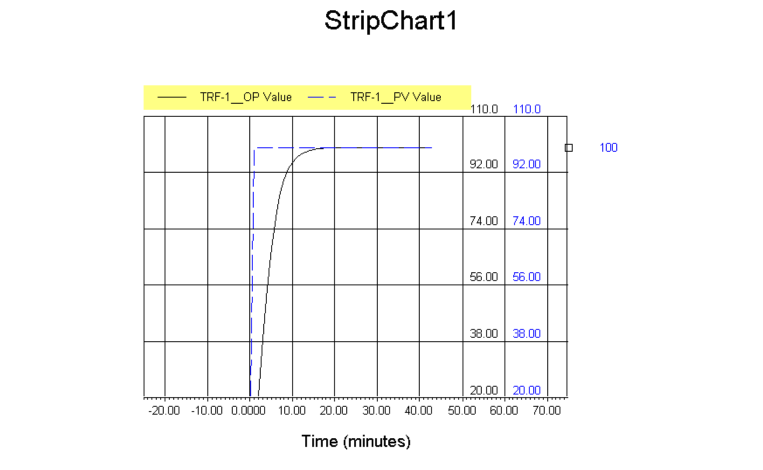
3. On the **2nd Order** page of the **Parameters** tab activate the 2nd Order check-box and enter the following parameters in the 2nd Order Lag Parameters group: K=1, T=2 minutes and x=1.
4. On the **Configuration** page of the **Parameters** tab enter a PV Range of 0 to 200 kgmole/h.
5. Start the Integrator. Change the molar flow rate of **Input** to 100 kgmole/h.

The PV range has no effect on the response of the second-order lag. However, it affects the Output range.

You would get the same response if you defined two first order lags with the parameters K=1 and T=2 minutes.

6. The response is shown below:

Figure 10.57



The form of the response is:

$$Flow(t) = K\left[1 - \left(1 + \frac{t}{T}\right)e^{-t/T}\right] \qquad (10.27)$$

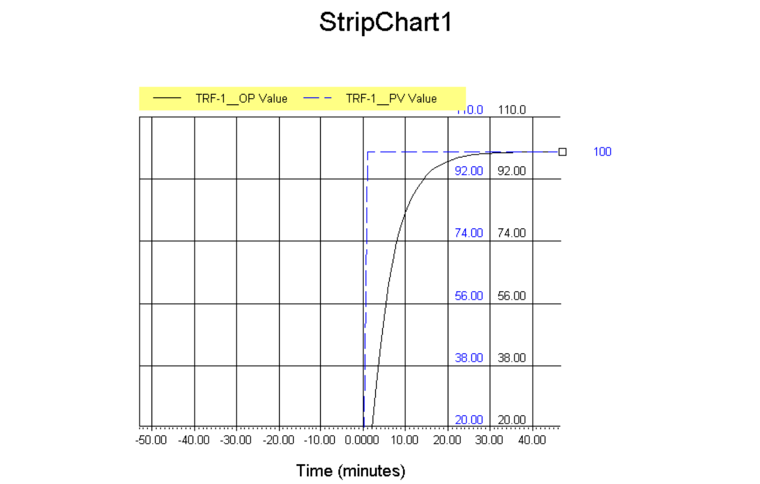
where: K = Process Gain

T = Time constant

In this case, the response will reach 26.4% of its final value in one time constant.

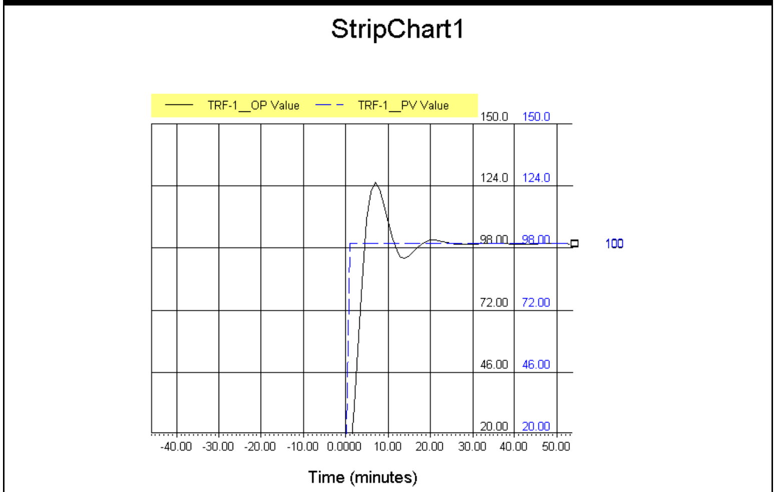
For an overdamped response, change ξ (**Xi**) to 1.5, reset the Integrator and the Input flow, and add the step change again:

Figure 10.58



Notice that the response does not overshoot the final value, but approaches it more slowly than the critically damped response. Finally, simulate an underdamped response by changing ξ (**Xi**) to 0.4:

Figure 10.59



In this case, notice that the response overshoots the final value and is oscillatory.

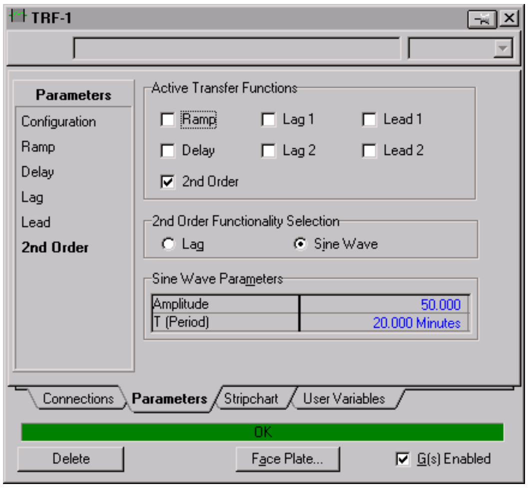
Note the significance of the K-value, or gain, of the second-order process. For a gain of 1.0, the output will change the same amount as the input. In this case, the input changed from 0-100 kgmole/h, and the output reached a lined-out value of 100 as well.

For a gain other than unity, the output will change the amount of the input change multiplied by the gain:

$$K \cdot \Delta Input = \Delta Output \cdot (@steadystate) \tag{10.28}$$

Sine Wave

Figure 10.60



The Sine Wave Transfer function is defined as follows:

$$G = \frac{K\omega}{s^2 + \omega^2} \tag{10.29}$$

where: ω = *Frequency of oscillation*

K = *Amplitude*

Note that the frequency is the inverse of the period ($\omega = 1/T$).

The Inverse Laplace of the Sine Wave Transfer Function is:

$$f(t) = K \sin \omega t \tag{10.30}$$

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 **Parameters** tab.

As usual, check the **Sine Wave** radio button on **2nd Order** page to simulate the Sine Wave. You cannot activate both the Standard 2nd Order and the Sine Wave at the same time.

Sine Wave Function Example

Suppose **SineOutput** were to feed a downstream unit operation. We wish to vary the flow of this stream sinusoidally by ±10% (between 90-110 kgmole/h).

1. Define the stream **Input** as shown earlier in this section, and set its flow to zero.
2. Create a new stream with the name **SineOutput**.
3. Install a Transfer Function Block with the following connections:

Tab [Page]	Input Area	Entry
Connections	Process Variable Source Object	Input
	Process Variable Source Variable	Molar Flow
	Transformed PV Target Object	SineOutput
	Transformed PV Target Variable	Molar Flow

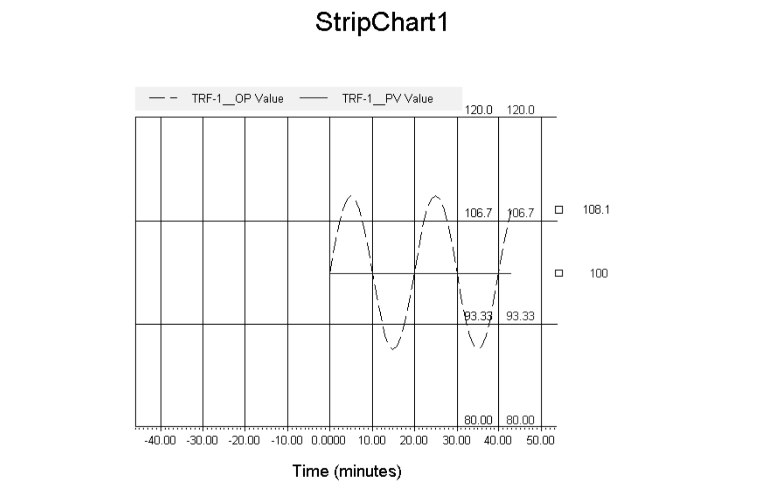
4. On the **2nd Order** page of the **Parameters** tab define the sinusoidal transfer function using the parameters K=50 and T = 20 minutes.
5. On the **Configuration** page of the **Parameters** tab enter a PV Range of 90 and 110 kgmole/h, respectively. These values define the range of the output - as mentioned earlier, regardless of the input or the Transfer Function parameters, the output will always stay in this range.

The K-value (transfer function gain) is:

$$K = \frac{Amplitude}{Range} \cdot 100 \tag{10.31}$$

In this case, K is 50, so the amplitude will be 50% of the range, which is 10 kgmole/h. Therefore, the output flow will rise 10 kgmole/h above and fall 10 kgmole/h below the input flow. This means that the output will oscillate between 90 and 110 kgmole/h, covering the entire range.

Figure 10.61

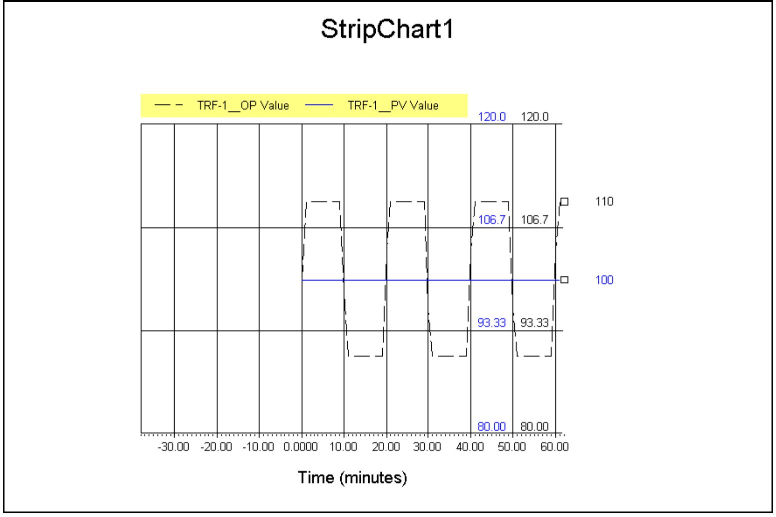


The sine wave completes one cycle every 20 minutes.

Square Wave Function Example

Now suppose that you wish to vary the output flow as a square wave function, with the same amplitude and frequency as the sine wave. Simply enter a large value for K, which will instruct HYSYS to produce a large-amplitude Sine wave. However, due to the Signal Minimum and Maximum values defined on the **Connections** tab, the output flow will only be allowed to reach a high value of 110 and a low value of 90 kgmole/h, thus producing a square wave. For example, try a K-value of 1000.

Figure 10.62



According to the equation for the K-value, the amplitude that the sine wave is trying to reach is:

$$\begin{aligned} Amplitude &= \frac{K \cdot Range}{100} \\ &= \frac{1000 \cdot 20}{100} \\ &= 200\ kgmol/hr \end{aligned}$$

Therefore, the wave is trying to reach a minimum of 100 - A = -100 kgmole/h and a maximum of 100 + A = 300 kgmole/h, but the Signal Min and Max values prevent it from travelling outside the 90-110 kgmole/h range. The wave rises rapidly because it must reach the large amplitude over the same period (20 minutes), in effect producing the Square wave function.

10.7 Controller Face Plate

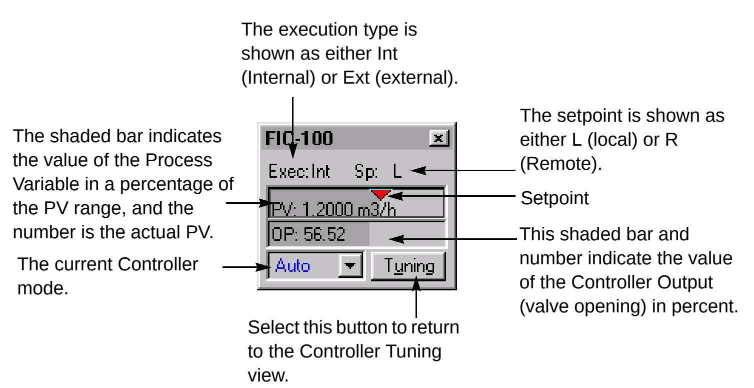
There are two ways that you can access the controller Face Plate: click the **Face Plate** button located at the bottom of the Controller's property view or by using the **Face Plate** option located in the **Tools** menu (see Section 7.7.1 - Face Plates of the User's Guide for more information).

Each controller's Face Plate varies in appearance, however the functionality remains the same. This section will provide 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 is displayed 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 display the execution type and the setpoint source.

Also note that you may change the mode of the Controller by selecting the drop-down menu at the bottom left of the Face Plate. The mode choices are identical to those on the **Parameters** tab. Clicking the **Tuning** button will return you to the **Tuning** tab of the Controller property view

Figure 10.63



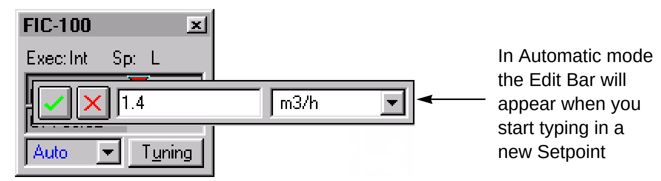
Changing the Setpoint and Output

You may 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 wish to change. For this example, the Setpoint (top field) will be changed. Start entering a new value for the SP, and HYSYS will provide an Edit Bar complete with the current default units. Once you have entered the value, press **ENTER** or select the check mark and HYSYS will accept the new Setpoint.

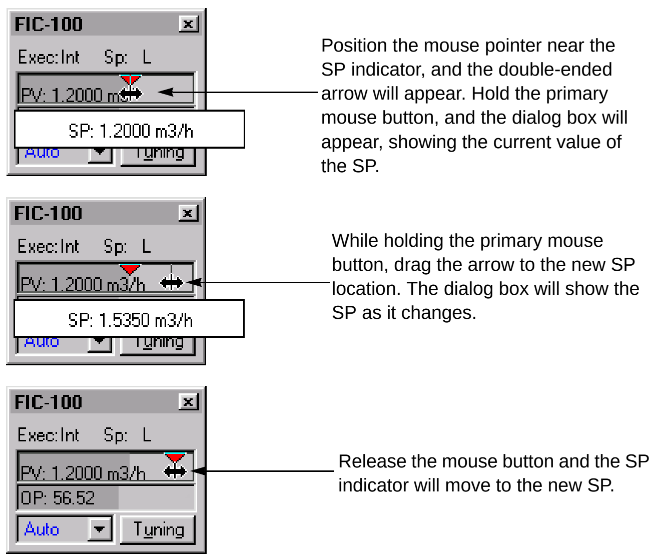
   Note that if you select an alternate unit, your value will be displayed in the faceplate using HYSYS display units. (See **Section 1.1.8 - The Property View** in **User's Guide** for a Description of how the Edit Bar works).

Figure 10.64



2. Place the mouse pointer near the red Setpoint indicator, and the cursor will change to a double-ended arrow. Hold down the primary mouse button, and a numerical dialogue box will appear below, showing the current value of the SP (in this case, 1.2 m^3/h). While holding the primary mouse button, drag the double-ended arrow to the new SP of 1.5350 m^3/h. The dialogue box will display the SP value as you drag. Release the primary mouse button to accept the new SP.

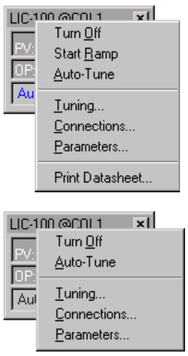
Figure 10.65



3. Place the mouse pointer at either end of the field, and the pointer will change to a single-ended arrow. Click the primary mouse button 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 will appear. Click the primary mouse button to increase the OP by 1%. Note that you may click the button consecutively to repeatedly increase (or decrease) the OP.

Object Inspection of Face Plates

Figure 10.66



The Object Inspection menu for a **Fixed Size** Face Plate is shown in Figure 10.66. The options associated with this menu are:

Refer to Chapter 11 - Control Theory of the ***Dynamic Modelling*** *manual for information on the object inspection options.*

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.

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.

10.8 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:

To set up a Strip Chart to track the PV and OP do the following:

*1. **Open** the DataBook 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.*

- 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*%.
- 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:

$$KU = \frac{4h}{\pi a} \tag{10.32}$$

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 will only work for systems with dead time.

11 Control Theory

Process control on a working level involves the control of variables such as flowrate, temperature, and pressure in a continuously-operating plant. Process control in a general sense attempts to maximize profitability, ensure product quality, and improve the safety and operability of the plant.

While steady state simulation in HYSYS allows the design engineer to optimize operating conditions in the plant, dynamic simulation allows you to:

- design and test a variety of control strategies before choosing one that may be suitable for implementation
- stress the system with disturbances as desired to test for plant performance

Even after a plant has started operation, process engineers may look for ways to improve the quality of the product, maximize yield, or reduce utility costs. Dynamic simulation using HYSYS allows the process engineer to compare alternative control strategies and operating schemes in order to improve the overall performance of the plant. In short, the engineer can accomplish a lot of analysis off-line on a dynamic simulator, instead of disturbing the actual process.

Three topics will be covered in this chapter. First, the characteristic parameters of a process will be discussed in the **Process Dynamics** section. Next, the control strategies available in HYSYS.Plant will be discussed in the **Controller Setup** section. Finally, the **General Guidelines** section will outline some steps you can follow to implement a control strategy in HYSYS.Plant. Included in this section are several techniques that may be used to determine possible initial tuning values for the controller operations.

11.1 Process Dynamics

As a precursor to understanding the concepts of process control, the dynamic characteristics of the process will be discussed. The task of designing a control scheme is best carried out if there is a good understanding of the process system being studied. A process' response to change can vary considerably depending on the manner in which the input is applied to the system and the nature of the system itself. Therefore, it is important to understand the dynamic characteristics of the process system before proceeding with the process control design.

As demonstrated in **Chapter 1 - Dynamic Theory**, many chemical engineering systems are non-linear in nature. However, it is convenient to define some essential characteristic parameters of a process system by approximating the system as linear.

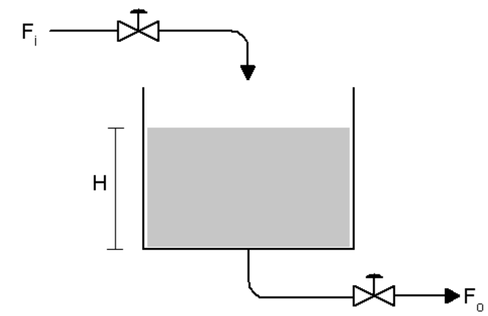
11.1.1 Characteristic Parameters of the Process System

It is easiest to define a chemical process system using the general conservation principle which state that:

$$\textit{Rate of accumulation} = \textit{Input} - \textit{Output} + \textit{Internal Generation} \qquad (11.1)$$

In order to describe some characteristic parameters of a chemical process system, the general conservation principle is applied to a flow relation first order liquid level system:

Figure 11.1



The conservation of material in the tank is expressed as follows:

$$A\frac{dH}{dt} = F_i - F_o \qquad (11.2)$$

where: H = *the liquid height in the tank*

A = *the cross-sectional area of the tank*

F_i = *the inlet flow rate*

F_o = *the exit flow rate*

There is a non-linear relationship describing the flow out of the bottom of the tank, F_o, and the liquid height in the tank, H. However, in order to express Equation (11.2) as a first order linear differential equation, it must be assumed that the exit flow varies linearly with height. Linearity can be assumed in situations in which the flow does not vary considerably over time. The exit flow, F_o, can be expressed in terms of the linearity constant, R (the valve resistance):

$$F_o = \frac{H}{R} \tag{11.3}$$

Equation (11.2) can therefore be expressed as:

$$A\frac{dH}{dt} = F_i - \frac{H}{R} \tag{11.4}$$

$$RA\frac{dH}{dt} + H = RF_i \tag{11.5}$$

Equation (11.5) is a general first-order differential which can be expressed in terms of two characteristic parameters: the steady-state gain, K, and the time constant, τ:

$$\tau\frac{dy}{dt} + y(t) = Ku(t) \tag{11.6}$$

where: y(t) = the output of the system

u(t)= input to the system

K = the steady-state gain

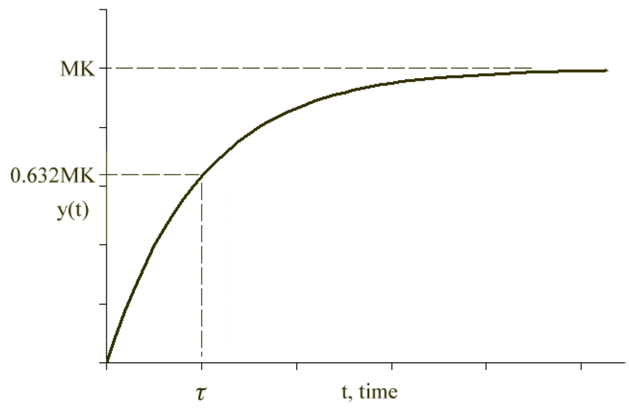
τ = the time constant of the system

The change in liquid level, H, is the change in the output of the system, y(t). The change in the input to the system, u(t), is the change in flow into the tank, F_i. Similarly, the time constant, τ, and the steady state gain, K, can be expressed as:

$$\tau = AR \text{ and } K = R \tag{11.7}$$

When a step function of magnitude M is applied to the general first-order system, the output response, y(t), is as follows:

Figure 11.2



First order response to a step input

As shown, the output, y(t), attains 63.2% of its final steady state value in one time constant. The output's response in equation form is:

$$y(t) = MK\left[1 - e^{\frac{-t}{\tau}}\right] \tag{11.8}$$

or in terms of the first-order tank example:

$$H(t) = MR\left[1 - e^{-\frac{t}{AR}}\right] \tag{11.9}$$

The following is a list of characteristic parameters that may be defined in terms of the first-order response illustrated in the previous example.

Process Gain

The process gain is defined as the ratio of the change in the process output to the change in the process input. The change in the process input is defined in **Equation (11.6)** as u(t). The change in the process output is defined as y(t). The first term in **Equation (11.6)** is transient and becomes zero at steady state. Therefore the gain can be calculated as shown in **Equation (11.10)**.

$$\text{Steady-state gain} = \frac{y(t)}{u(t)} = K \tag{11.10}$$

For this liquid level example, the steady state gain, K, is the valve resistance, R. Therefore, a step change in the flow into the tank of magnitude M will result in a change in liquid level, H(t), in the tank equal to MR.

Time Constant

The time constant, τ, defines the speed of the response. The response of the system will always follow the profile shown in **Figure 11.2**. After τ time units, the response y(t) equals 0.632MK or 63.2% of the ultimate gain. This will always be true for first-order systems. For this liquid level example, the time constant is the product of the area of the tank, A, and the resistance of the exit valve, R.

Capacity

Definition 1

Capacity can be defined simply as the volume or storage space of a system. The capacitance of a system dampens the output causing the response to take time to reach a new steady state. For electrical systems, the capacity is defined in terms of the resistance of the system and the time constant of the response:

$$C = \frac{\tau}{R} \tag{11.11}$$

In the liquid level example, the capacity is the cross sectional area of the tank. Since the capacity of a system is proportional to the time constant, τ, it can be concluded that the larger the capacity, the slower the response of the system for a given forcing function.

In first order systems, the capacity of a system has no effect on the process gain. However, the capacity varies in direct proportion with the time constant of a system.

Definition 2

A system's capacity is also defined as its ability to attenuate an incoming disturbance. Attenuation is defined as:

$$\text{Attenuation} = 1 - \frac{\text{Response Amplitude out of the system}}{\text{Disturbance Amplitude into the system}} \tag{11.12}$$

$$\text{Attenuation} = 1 - \text{Amplitude Ratio}$$

The input, u(t), to the first order system can be changed from a step function to a sinusoidal function:

$$u(t) = M\sin(\omega t) \tag{11.13}$$

where: ω *= the rotational speed of the input response.*

M *= the amplitude of the input function*

The response of the system y(t) becomes:

$$y(t) = MK\left[\frac{\omega\tau}{(\omega\tau)^2+1}e^{-t/\tau} + \frac{1}{\sqrt{(\omega\tau)^2+1}}\sin(\omega t+\phi)\right] \tag{11.14}$$

where: $\phi = \tan^{-1}(-\omega\tau)$

After the transient term becomes negligible, (the first bracketed term), an ultimate periodic response remains (the second term). The response amplitude of the system is therefore:

$$y(t) = \frac{MK}{\sqrt{(\omega\tau)^2+1}} \tag{11.15}$$

Since the disturbance amplitude into the system is M, the amplitude ratio is:

$$AR = \frac{K}{\sqrt{(\omega\tau)^2+1}} \tag{11.16}$$

Dead Time

While capacitance is a measure of how fast a system responds to disturbances, dead time is a measure of the amount of time that elapses between a disturbance to the system and the observed response in the system. Time delays in a system can become significant depending on the nature of the process and the location of measuring devices around the process. It is usually the time associated with the transport of material or energy from one part of the plant to another that contributes to time delays observed in a system. The dead time of a process may be easily modelled using the **Transfer Function** block operation.

11.2 Basic Control

The PID Controller operation is the primary tool that you can use to manipulate and control process variables in the dynamic simulation. You can implement a variety of feedback control schemes by modifying the tuning parameters in the **PID Controller** operation. Tuning parameters can be modified to incorporate proportional, integral, and derivative action into the controller. A **Digital On/Off** control operation is also available. Cascade control may be modelled using interacting **PID Controller** operations. Feedforward control can be implemented into the simulation model using the **Spreadsheet** operation.

Instrumentation dynamics can also be modelled in HYSYS.Plant, increasing the realism of the simulation with real valve dynamics. Final control elements can be modelled with hysteresis. The valve response to controller input can be modelled as instantaneous, linear, or first order. Dead time, lags, leads, whether they originate from disturbances or within the process control loop may be modelled effectively using the **Transfer function** operation.

11.2.1 Terminology

Before reviewing the major control operations that are available in HYSYS, it is useful to describe some terms.

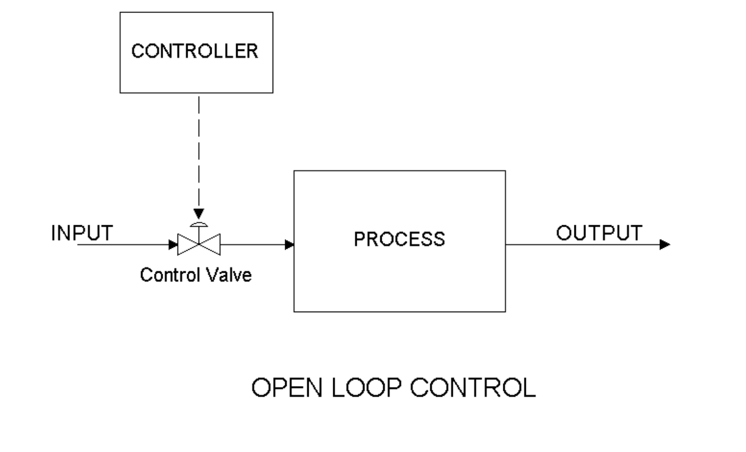
Disturbances

A disturbance upsets the process system and causes the output variables to move from their desired set points. Disturbance variables cannot be controlled or manipulated by the process engineer. The control structure should account for all disturbances that can significantly affect a process. The disturbances to a process can either be measured or unmeasured.

Open Loop Control

An open loop response from a process is determined by varying the input to a system and measuring the output's response. The open-loop response to a first-order system from a step input is shown in Figure 11.2. In open-loop control, the controller sets the input to the process without any knowledge of the output variable that closes the loop in feedback control schemes.

Figure 11.3

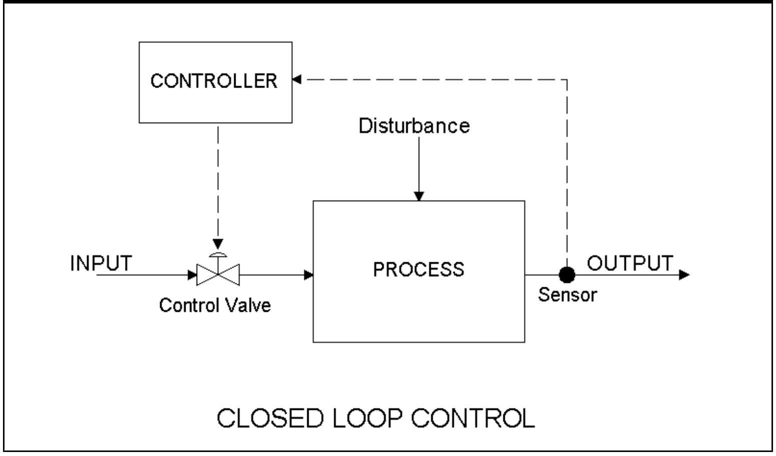


A common example of open-loop control is the control of traffic in a city. The traffic lights change according to a set of predetermined rules.

Feedback Control (Closed Loop)

Feedback control is achieved by "feeding back" process output information to the controller. The controller makes use of the current information about the process variable in order to determine what action to take to regulate the process variable. This is the simplest and most widely used control structure in chemical process systems.

Figure 11.4

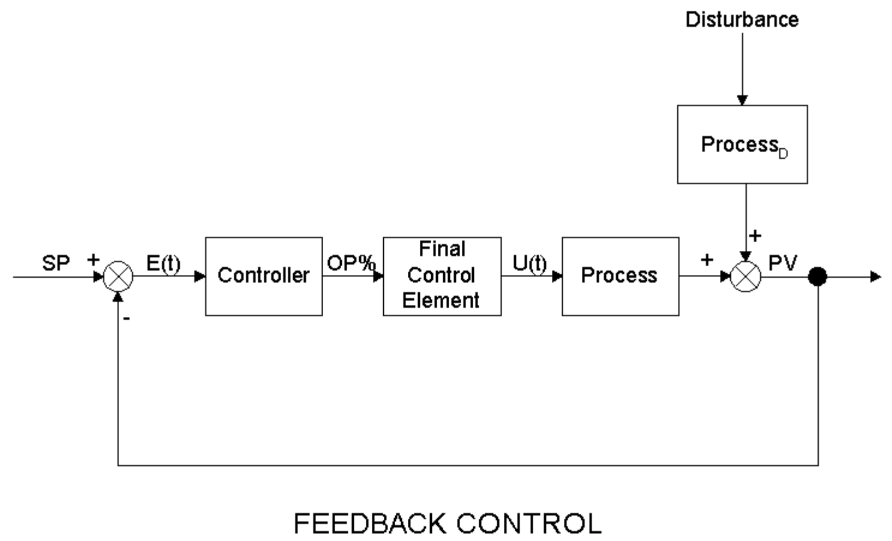


Feedback control attempts to maintain the output variable, PV, at a user-defined set point, SP. There are some basic steps that are carried out by the controller in order to achieve this task:

1. Measure the output variable, PV.
2. Compare the measured value, PV, with the desired set point value, SP. Calculate the error, E(t), between the two values. The definition of error depends on the whether the controller is direct or reverse-acting.
3. Supply the error, E(t), to the general control equation. The value of the desired percent opening of the control valve, OP%, is calculated.
4. The value of OP% is passed to the final control element which determines the input to the process, U(t).

5. The entire procedure is repeated.

Figure 11.5



The general control equation for a PID controller is given by:

$$OP(t) = K_c E(t) + \frac{K_c}{T_i}\int E(t) + K_c T_d \frac{dE(t)}{dt} \tag{11.17}$$

where: OP(t) = Controller output at time t

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

Direct and Reverse Acting

The input to the feedback controller is called the error or the difference between the output process variable and the set point. The error is defined differently depending on whether the process has a positive or negative steady state gain. For a process with a positive steady state gain, the error should be defined as reverse-acting.

$$E(t) = SP(t) - PV(t) \tag{11.18}$$

where: SP(t) = set point

PV(t) = measured output process variable

That is, if the PV *rises* above the SP, the OP, or input to the process, *decreases*. If the PV *falls* below the SP, the OP *increases*.

For a process with a negative steady state gain, the error should be set as direct acting:

$$E(t) = PV(t) - SP(t) \tag{11.19}$$

That is, if the PV *rises* above the SP, the OP, or input to the process, *increases*. If the PV *falls* below the SP, the OP *decreases*.

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. See **Chapter 10 - Logical Operations** in this manual for some other examples of direct and reverse-acting controllers.

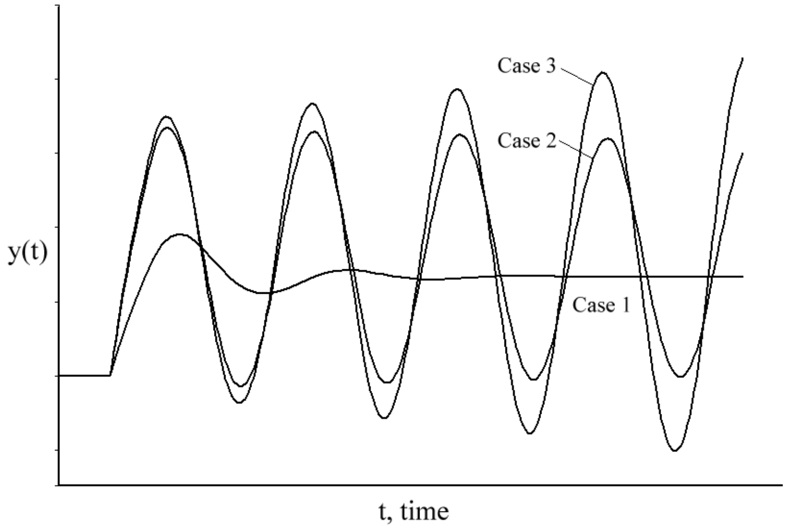
Stability

The stability of a system is a very important aspect to consider when designing control schemes. Many systems have oscillatory responses, depending on its controller tuning parameters. When a process is upset by a *bounded* disturbance or bounded change in the input forcing function, the output typically will respond in one of three ways:

1. The response will proceed to steady state and stabilize.
2. The response will oscillate continuously with a constant amplitude.

3. The response will grow continuously and never reach steady state conditions.

Figure 11.6



Stability Response Cases

The system is generally considered stable if the response proceeds to a steady state value and stabilizes. It is considered unstable if the response continues to fluctuate. A stable *open-loop* response is said to be *self-regulating*. If the open loop response of a system is not stable, it is said to be *non-self-regulating*. For instance, a pure integrating process, such as a tank with a pumped (constant) exit flow, is non-self-regulating since a bounded increase in the flow input to the system from steady state will result in the response (liquid height) to increase continuously.

A prerequisite for *closed-loop* control is that the closed-loop response is stable. The closed-loop response can vary considerably depending on the tuning parameters used in the feedback control equation. In general, a higher controller gain gives tighter control. However, the value of K_c cannot increase indefinitely. The response will remain stable up to a certain value of K_c. Increasing K_c beyond the stability limit will cause the closed-loop response to become unstable.

A number of factors can affect the stability of a closed-loop system:

- Tuning parameters
- Non-linearities in the process
- Range and non-linearities in the instruments
- Interactions between control loops
- Frequency of disturbance
- Capacity of process
- Noise in measurement of process variables

11.2.2 Available Control Operations

Modelling Hardware Elements

The plant may be simulated more accurately by modelling the hardware elements of the control loop. Non-linearities may be modelled in the **VALVE** operation in the **Actuator** page of the **Dynamics** tab.

Sensors

Sensors are used to measure process variables. In HYSYS, the sensing instrument is incorporated directly in the **PID Controller** operation. You can choose the range of the sensing instrument in the Min and Max PV parameters in the controller operation. It is assumed in HYSYS that the PID controller is perfectly accurate in its measurement of the process variable.

Final Control Element- Valve Dynamics

You have the option of specifying a number of different dynamic modes for the valve. If valve dynamics are very quick compared to the process, the instantaneous mode may be used. The following is a list of the available dynamic modes for the **VALVE** operation:

Valve Mode	Description
Instantaneous	In this mode, the actuator moves instantaneously to the desired OP% position from the controller.
First Order	A first order lag can be modelled in the response of the actuator position to changes in the desired OP%. The actuator time constant can be specified in the **Parameters** field. Similarly, a first order lag can be modelled in the response of the actual valve position to changes in the actuator position. The valve stickiness time constant is specified in the **Parameters** field. In effect, a second order lag can be modelled between the valve position and the desire OP%.
Linear	The actuator can be modelled to move to the desire OP% at a constant rate. This rate is specified in the **Parameters** field

For more information regarding the dynamic **VALVE** operation, see **Section 5.2 - Valve** in **Chapter 5 - Piping Equipment** of this manual.

Final Control Element- Valve Type

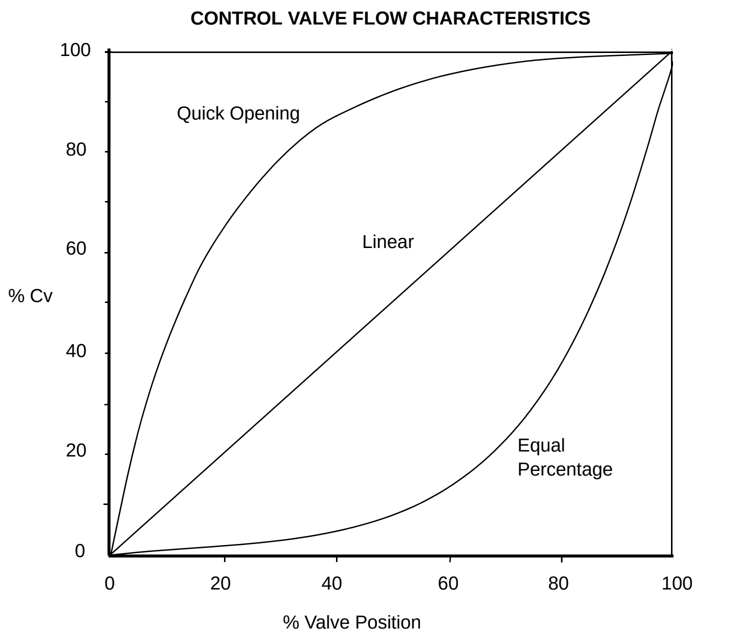
The flowrate through a control valve varies as a function of the valve percent opening and the **Valve Type**. Valve type may be defined more easily by expressing flow as a percentage, C_v (0% representing no flow conditions and 100% representing maximum flow conditions). The valve type can then be defined as the dependence on the quantity of $\%C_v$ as a function of the actual valve percent opening.

There are three different valve characteristics available in HYSYS. The valve types are specified in the **Ratings** tab in the **Valve Type and Sizing Methods** group.

Valve Type	Description
Linear	A control valve with linear valve characteristics has a flow which is directly proportional to the valve % opening.
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.
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 valve characteristics are shown graphically as follows:

Figure 11.7



Feedback Control

Digital On/Off

Digital On/Off control is one of the most basic forms of regulatory control. In HYSYS, it is implemented using the **DIGITAL POINT** operation. An example of On/Off control is a home heating system. When the thermostat detects that the temperature is below the set point, the heating element turns on. When the temperature rises above the setpoint, the heating element turns off.

Control is maintained using a switch as a final control element (FCE). On/Off control parameters are specified in the **Parameters** page of the **DIGITAL POINT** operation in HYSYS. If the **OP is ON** option is set to "PV < Threshold", the controller output turns on when the PV falls below the setpoint.

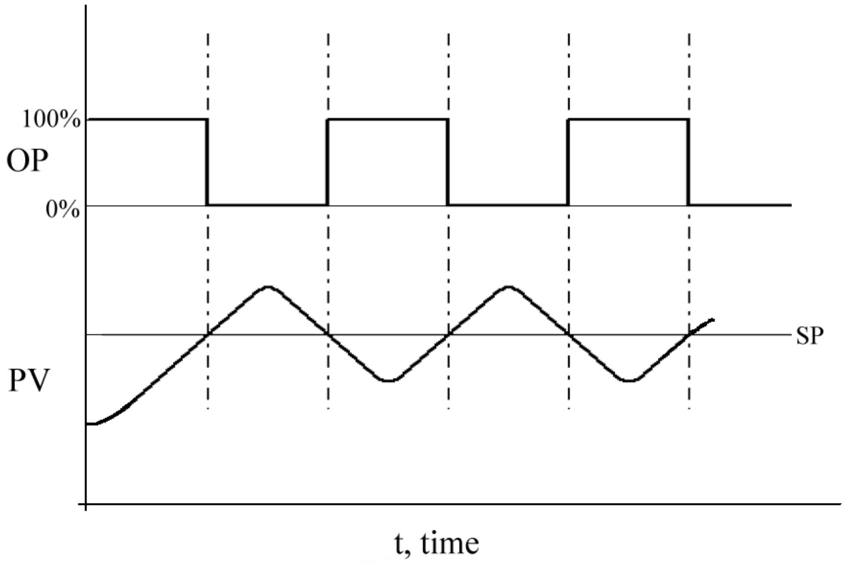
$$OP = 0\% \text{ for PV} > \text{SP and } OP = 100\% \text{ for PV} < \text{SP} \tag{11.20}$$

The opposite is true when the **OP is ON** option is set to "PV > Threshold".

$$OP = 0\% \text{ for PV} < \text{SP and } OP = 100\% \text{ for PV} > \text{SP} \tag{11.21}$$

One main characteristic of the On/Off controller is that the PV will always cycle about the setpoint.

Figure 11.8



On-Off Controller Response

The cycling frequency will depend on the dynamics of the process. Those systems with a large capacity (large time constant) will cycle less frequently. The On/Off controller is an appropriate controller if the deviation from the setpoint is within an acceptable range and the cycling does not destabilize the rest of the process.

For more information on the **DIGITAL POINT** operation in HYSYS, see **Chapter 10 - Logical Operations**.

Proportional Control (P-only)

Unlike On/Off control, proportional control can damp out oscillations from disturbances and stop the cycling of the process variable. P-only control is implemented in HYSYS by setting the values of T_d and T_i to zero in the **PID Controller** operation. With P-only control, oscillations that occur in the process variable due to disturbances or changes in the setpoint dampen out the quickest (have the smallest natural period) among all other simple feedback control schemes. The output of the proportional control is defined as:

$$OP(t) = OP_{ss} + K_c E(t) \tag{11.22}$$

The value of the bias, OP_{ss}, is calculated when the controller is switched to **Automatic** mode. The setpoint is defaulted to equal the current PV. In effect, the error becomes zero and OP_{ss} is then set to the value of OP(t) at that time.

A sustained offset between the process variable and the setpoint will always be present in this sort of control scheme. The error becomes zero only if:

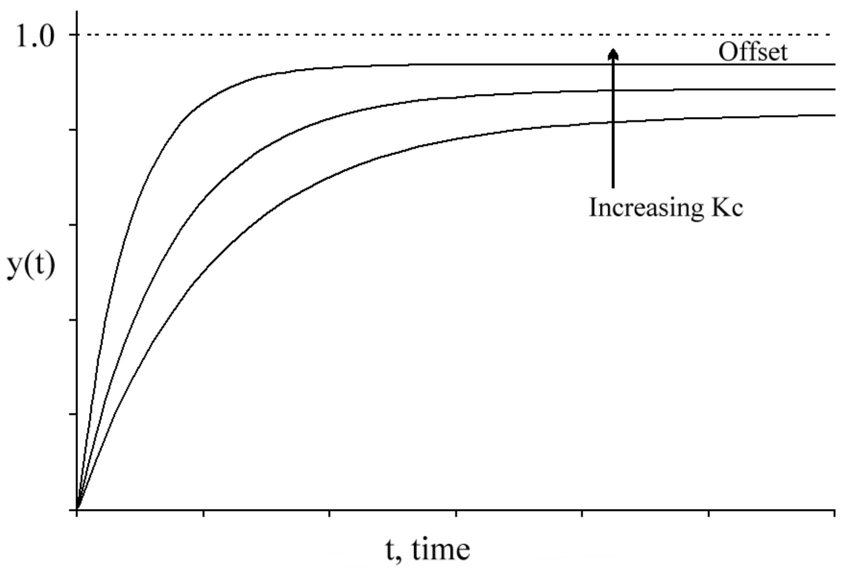
- the bias, OP_{ss}, equals the operating variable, OP
- K_c becomes infinitely large

However, K_c cannot practically become infinitely large. The magnitude of K_c is restricted by the stability of the closed-loop system.

In general, a higher controller gain gives tighter control. However, the value of K_c cannot increase indefinitely. The response will remain stable up to a certain value of K_c. Increasing K_c beyond the stability limit will cause the closed-loop response to become unstable.

The following shows the effect of the magnitude of K_c on the closed loop response of a first order system to a unit step change in setpoint.

Figure 11.9



Closed loop response of a system under P-Only Control

Proportional only control is suitable when a quick response to a disturbance is required. P-only control is also suitable when steady-state offsets are unimportant, or when the process possesses a large integrating process (has a large capacity). Many liquid level control loops are under P only control. If a sustained error is undesirable, integral action is required to eliminate the offset.

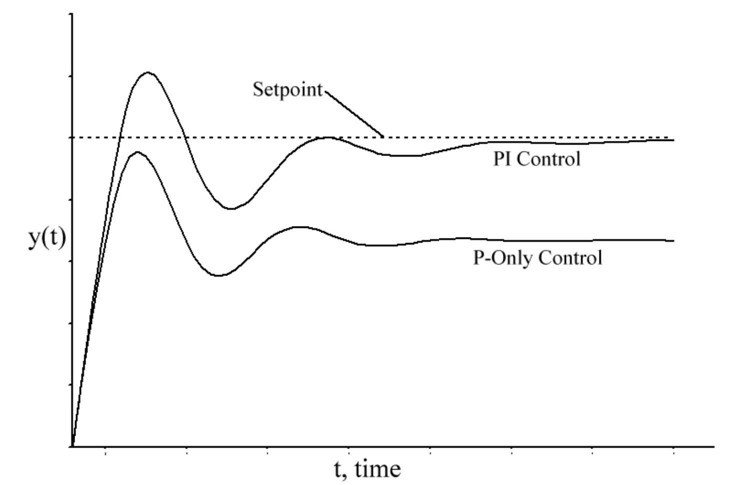
Proportional + Integral Control (PI)

Unlike P-only control, proportional + integral control can dampen out oscillations and return the process variable to the setpoint. Despite the fact that PI control results in zero error, the integral action of the controller increases the natural period of the oscillations. That is, PI control will take longer to line out (dampen) the process variable than P-only control. The output of the proportional controller + integral controller is defined as:

$$OP(t) = K_c E(t) + \frac{K_c}{T_i}\int E(t) \tag{11.23}$$

The integral term serves to bring the error to zero in the control scheme. The more integral action there is, the slower the response of the controller will be. The integral term continuously moves to eliminate the error. The closed loop response of a process with PI control and P-only control is shown as follows:

Figure 11.10



Proportional and PI Control

The integral time, T_i, is defined as the amount of time required for the controller output to move an amount equivalent to the error. Because the relationship between T_i and the control action is reciprocal, increasing T_i will result in less integral action, while decreasing T_i will result in greater integral action. The integral time should be decreased (increased integral action) just enough to return the process variable to the setpoint. Any more action will only serve to lengthen the response time.

PI control is suitable when offsets cannot be tolerated. The majority of controllers in chemical process plants are under PI control. They combine accuracy (no offset) with a relatively quick response time. However, the added integral action acts as a destabilizing force which may cause oscillations in the system and cause the control system to become unstable. The larger the integral action the more likely it will become unstable.

Proportional Integral Derivative Control (PID)

If the response of a PI controller to a disturbance is not fast enough, the derivative action in a PID controller can reduce the natural period of oscillations even further. By measuring the rate of change in error, the controller can anticipate the direction of the error and thus respond more quickly than a controller without derivative action. The output of the proportional + integral + derivative controller is defined as:

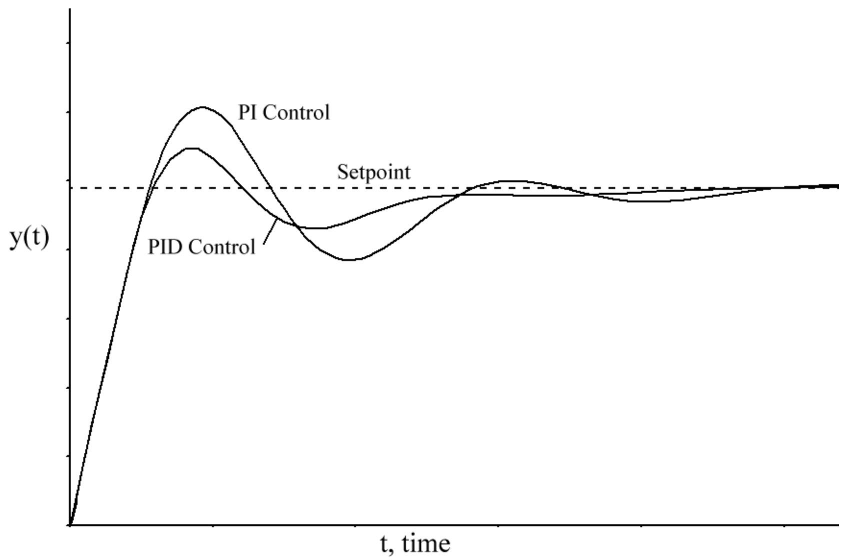
$$OP(t) = K_c E(t) + \frac{K_c}{T_i}\int E(t) + K_c T_d \frac{dE(t)}{dt} \tag{11.24}$$

T_d is defined as the time required for the proportional action to reach the same level as the derivative action. It is, in effect, a lead term in the control equation. For a ramped input, the proportional only response will be ramped, as well. For the same ramped input the derivative only response will be constant.

As the slope of the measured error increases to infinity, so does the derivative action. While a perfect step change with a slope of infinity in either the setpoint or the measured process variable is not physically possible, signals which have short rise times and fall times can occur. This adversely affects the output of the derivative term in the control equation, driving the controller response to saturation.

Derivative action control is best for processes which have little or no dead times and large capacities. Processes such as these having large lags benefit from the additional response speed that derivative action provides. While the integral term in PID control schemes reduces the error to zero, it also adds a considerable lag to the response compared to P-only control. It is the derivative action in PID control which shortens the controller's response to be comparable to the response of a P-only controller. However, if a controller has a very noisy input which cannot be filtered or minimized in the process, PID control is not a suitable control scheme.

Figure 11.11



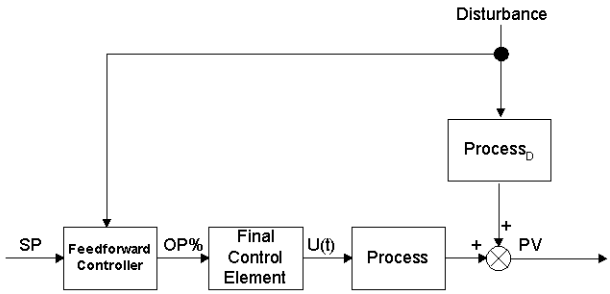
PI and PID Control

Feedforward Control

Feedforward control may be used in cases for which feedback control cannot effectively control a process variable. The main disadvantage of feedback control is that the controller must wait until disturbances upset the process before responding. With feedforward control, the controller can compensate for disturbances *before* the process is affected. Cascade control is useful when measured disturbances significantly affect the input to a process. On the other hand, feedforward control is useful if there are measured disturbances which affect the output of the process.

With feedback control, the controller requires information about the controlled process variable, PV, and the setpoint, SP, in order to determine the value of OP%, the desired valve percent opening of the input to the process. In order to determine the value of OP%, the feedforward controller requires information from two variables: the setpoint of the process variable, SP, and the disturbance affecting the process. A steady-state process model is used in the feedforward controller to determine the value of OP%.

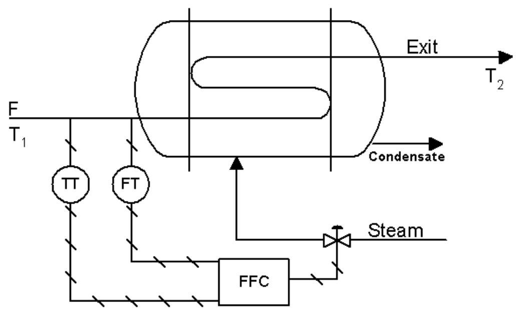
Figure 11.12



FEEDFORWARD CONTROL

Consider an example of a liquid stream being heated in a steam heat exchanger.

Figure 11.13



Feedforward Control of a Heat Exchanger

It is desired to control the Exit stream temperature, T_2, at a certain setpoint, SP, using the Steam flow as the manipulated variable. However, the process suffers from frequent changes in the Feed temperature, T_1. In order to determine the value of OP%, the values of SP and T_1 are required by the controller. At steady-state, the overall energy balance relates the steam flow to the disturbance of the process, T_1, and the temperature of stream Exit, T_2:

$$F_s\lambda - FC_p(T_2 - T_1) = 0 \qquad (11.25)$$

where: F_s *= the steam flow*

λ *= the heat of condensation for steam*

F *= the flow of stream Exit*

C_p *= the specific heat of stream Exit*

From this process model, the desired value of steam flow into the heat exchanger can be calculated. The flow of steam must be calculated such that the temperature of stream Exit, T_2, equals the desired temperature, SP. Therefore, **Equation (11.25)** becomes:

$$F_s = \frac{Cp}{\lambda}F(SP - T_1) \qquad (11.26)$$

In order to calculate the feedforward controller output, a linear relation is assumed to exist between the steam flow and the valve opening of the steam valve. Therefore the final form of the feedforward controller equation is:

$$OP(t) = \frac{C_p}{\lambda}F(SP - T_1)\frac{steam\ valve\ span}{100\%} \qquad (11.27)$$

There are some points to consider in order to successfully implement a feedforward control system:

1. It cannot be implemented if the disturbance is not measurable. If unexpected disturbances enter the process when pure feedforward control is used, no corrective action is taken and the errors will build up in the system.
2. A fairly accurate model of the system is required.

3. The feedforward controller contains the reciprocal of the process model. Even if the process model is accurate, a time delay in the process model implies that a predictor is required in the feedforward controller. Unfortunately, it is impossible to predict the nature of disturbances before they occur.

It is important to note that the process variable to be controlled is not measured using feedforward control. There is no way of confirming that the process variable is attenuating disturbances or maintaining a desired setpoint. Considering that an accurate model of the process is usually not available, that the process or valve dynamics are not accounted for in this control scheme, and that the valve opening percent is not related linearly to the flow in most dynamic simulation applications, there will probably be an offset between the actual controlled variable and its desired setpoint. Therefore, feedback control is often used in conjunction with feedforward control to eliminate the offset associated with feedforward-only control.

Feedforward control in HYSYS, can be implemented using the **Spreadsheet** operation. Variables can be imported from the simulation flowsheet. A feedforward controller can be calculated in the spreadsheet and the controller output exported to the main flowsheet. If the operating variable, OP, is a valve in the plant, the desired controller output calculated by the **Spreadsheet** should be exported to the *Actuator Desired Position* of the valve.

11.3 Advanced Control

11.3.1 Model Predictive Control

Model Predictive Control (MPC) refers to a class of algorithms that compute a sequence of manipulated variable adjustments in order to optimize the future behaviour of a plant. A typical MPC has the following capabilities:

1. Is capable of handling multivariable systems with process interactions.
2. Encapsulates the behaviour of multiple Single Input Single Output (SISO) controllers and de-couplers.
3. Utilizes a process model, i.e., a first order model or a step response data is required.
4. Incorporates the features of feedforward control, i.e., must be measured disturbance by taking in consideration the model disturbances in its predictions.
5. Is posed as an optimization problem and is therefore capable of meeting the control objectives by optimizing the control effort, and at the same time is capable of handling constraints.

MPC technology was originally developed to meet the specialized control needs of power plants and petroleum refineries, but it can now be found in a wide variety of application areas including chemicals, food processing, automotive, aerospace, metallurgy, and pulp and paper.

MPC versus PID

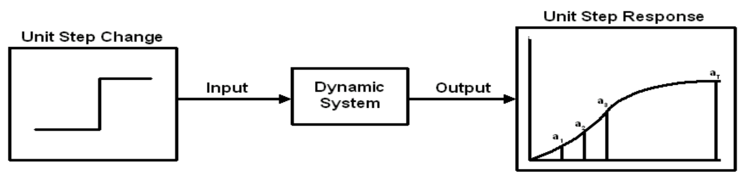
When the MPC controller is compared to the ubiquitous PID controller some differences are readily apparent. First, it is essential that there exist a model of the process in order to use a MPC controller. Like most advanced controllers, the model of the process is used to form predictions of the process outputs based on present and past values of the input and outputs. This prediction is then used in an optimization problem in which the output is chosen so that the process will reach or maintain its setpoint at some projected time in the future.

MPC Theory

Currently most model predictive control techniques like Dynamic Matrix Control (DMC) and Model Algorithmic Control (MAC) are based on optimization of a quadratic objective function involving the error between the setpoint and the predicted outputs. In these cases, a discrete impulse response model can be used to derive the objective function.

Let a_0, a_1, a_2,...,a_T represent the value of the unit step response function obtained from a typical open-loop process, as shown in **Figure 11.14**:

Figure 11.14



Implementation of a Unit Step Change to a Dynamic System

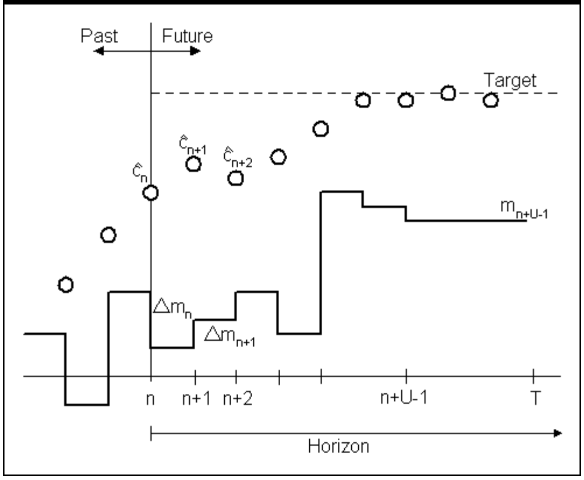
From the above figure, you can define $a_i = 0$ for $i \le 0$. Consider a step response resulting from a change Δm in the input. Let c_n be the actual output, $\hat{c}_n$ the predicted value of the output variable and m_n the value of the manipulated variable at the *n*th sampling interval. If there is no modelling error and no disturbances.

$$\hat{c}_n = c_n \tag{11.28}$$

Denoting $\Delta m_i = m_i - m_{i-1}$, the convolution model of the single step response function, see **Figure 11.15**, is given as follows:.

$$\hat{c}_{n+1} = c_0 + \sum_{i=1}^{T} a_i \Delta m_{n+1-i} \tag{11.29}$$

Figure 11.15



The control horizon U is the number of control actions (or control moves) that are calculated in order to affect the predicted outputs over the prediction horizon V, i.e., over the next V sampling periods. Similarly, the discrete model can be written as

$$\hat{c}_{n+1} = c_0 + \sum_{i=1}^{T} h_i m_{n+1-i} \tag{11.30}$$

Where h_i is the impulse response coefficient. Shifting the model back one time step, **Equation (11.30)** can be written as

$$\hat{c}_n = c_0 + \sum_{i=1}^{T} h_i m_{n-i} \tag{11.31}$$

Subtracting **Equation (11.31)** from **Equation (11.30)**, a recursive form of the model expressed in incremental change Δm can be obtained:

$$\hat{c}_{n+1} = \hat{c}_n + \sum_{i=1}^{T} h_i \Delta m_{n+1-i} \tag{11.32}$$

In order to provide corrections for the influence of model errors and unmeasured load changes during the previous time step, a corrected prediction c^*_{n+1} is utilize in the model. This corrected value is obtained by comparing the actual value of c_n with $\hat{c}_n$ and then shifting the correction forward, as follows:

$$c^*_{n+1} - \hat{c}_{n+1} = c_n - \hat{c}_n \tag{11.33}$$

Substituting the corrected prediction in **Equation (11.32)** results in the following recursive form:

$$c^*_{n+1} = c_n + \sum_{i=1}^{T} h_i \Delta m_{n+1-i} \tag{11.34}$$

The **Equation (11.34)** can be extended to incorporate predictions for a number of future time steps allowing the model-based control system to anticipate where the process is heading. A new design parameter called prediction horizon *V* is introduced, which influences control system performance, and is expressed in terms of incremental changes in the manipulated variable:

$$c^*_{n+j} = c^*_{n+j-1} + \sum_{i=1}^{T} h_i \Delta m_{n+j-i} \tag{11.35}$$

where:

j = 1, 2, ..., V

Suppose that arbitrary sequence of U input changes are made, then using a prediction horizon of V sampling periods **Equation (11.35)** can be expressed in a vector-matrix form as,

$$\begin{bmatrix} c^*_{n+1} \\ c^*_{n+2} \\ c^*_{n+3} \\ \cdot \\ \cdot \\ \cdot \\ c^*_{n+V} \end{bmatrix} = \begin{bmatrix} a_1 & 0 & 0 & \dots & 0 \\ a_2 & a_1 & 0 & & 0 \\ a_3 & a_2 & a_1 & & 0 \\ \cdot & & & & \cdot \\ \cdot & & & & \cdot \\ \cdot & & & & \cdot \\ a_V & a_{V-1} & a_{V-2} & \dots & a_{V-U+1} \end{bmatrix} \begin{bmatrix} \Delta m_n \\ \Delta m_{n+1} \\ \Delta m_{n+2} \\ \cdot \\ \cdot \\ \cdot \\ \Delta m_{n+U-1} \end{bmatrix} + \begin{bmatrix} c_n + P_1 \\ c_n + P_2 \\ c_n + P_3 \\ \cdot \\ \cdot \\ \cdot \\ c_n + P_V \end{bmatrix} \tag{11.36}$$

where:

$$a_i = \sum_{j=1}^{i} h_j$$

and

$$P_i = \sum_{j=1}^{i} S_j \qquad \text{for} \qquad i = 1, 2, .., V \tag{11.37}$$

$$S_i = \sum_{i=j+1}^{T} h_i \, \Delta m_{n+j-i} \qquad \text{for} \qquad j = 1, 2, .., V \tag{11.38}$$

Using the predicted behaviour of the process (see **Equation (11.36)**) over the prediction horizon, a controller in model predictive control can be designed. The control objective is to compute the manipulated variables to ensure that the predicted response has certain desirable characteristics i.e. to have the corrected predictions c^*_{n+j} approach the set point as closely as possible. One sampling period after the application of the current control action, the predicted response is compared with the actual response. Using the corrective feedback action for any errors between actual and predicted responses, the entire sequence of calculation is then repeated at each sampling instant. Denoting the setpoint trajectory, i.e., the desired values of the setpoint V time steps into the future, as r_{n+j}, $j = 1, 2, \ldots, V$, **Equation (11.36)** can be written as:

$$\hat{\mathbf{E}} = -\mathbf{A}\,\Delta\mathbf{m} + \hat{\mathbf{E}}' \tag{11.39}$$

Where $\mathbf{A}$ is the $V \times U$ triangular matrix and $\Delta\mathbf{m}$ is the $U \times 1$ vector of future control moves. $\hat{\mathbf{E}}$ and $\hat{\mathbf{E}}'$ are referred to as closed loop and open loop predictions, respectively, and are define as follows:

$$\hat{\mathbf{E}} = \begin{bmatrix} r_{n+1} - c^*_{n+1} \\ r_{n+2} - c^*_{n+2} \\ . \\ . \\ . \\ r_{n+V} - c_{n+V} \end{bmatrix} \qquad \hat{\mathbf{E}}' = \begin{bmatrix} E_n - P_1 \\ E_n - P_2 \\ . \\ . \\ . \\ E_n - P_V \end{bmatrix}$$

For a perfect match between the predicted output trajectory of the closed-loop system and the desired trajectory, then $\hat{\mathbf{E}} = 0$ and **Equation (11.39)** becomes:

$$\Delta\mathbf{m} = (\mathbf{A})^{-1}\hat{\mathbf{E}}' \tag{11.40}$$

The best solution can be obtained by minimizing the performance index,

$$J[\Delta \mathbf{m}] = \hat{\mathbf{E}}^T \hat{\mathbf{E}} \tag{11.41}$$

Here the optimal solution for an over determined system $(U < V)$ turns out to be the least squares solution and is given by

$$\Delta \mathbf{m} = (\mathbf{A}^T \mathbf{A})^{-1} \mathbf{A}^T \hat{\mathbf{E}}' = \mathbf{K}_c \hat{\mathbf{E}}' \tag{11.42}$$

Where $(\mathbf{A}^T \mathbf{A})^{-1} \mathbf{A}^T$ is the pseudo-inverse matrix and $\mathbf{K}_c$ is the matrix of feedback gains (with dimensions $V \times U$).

One of the shortcomings of **Equation (11.41)** is that it can result in excessively large changes in the manipulated variable, when $\mathbf{A}^T \mathbf{A}$ is either ill-conditioned or singular. One way to overcome this problem is by modifying the performance index by penalizing movements of the manipulated variable.

$$J[\Delta \mathbf{m}] = \hat{\mathbf{E}}^T \Gamma_{\boldsymbol{u}} \hat{\mathbf{E}} + \Delta \mathbf{m}^T \Gamma_y \Delta \mathbf{m} \tag{11.43}$$

Where $\Gamma_{\boldsymbol{u}}$ and Γ_y are positive-definite weighting matrices for predicted errors and control moves, respectively. These matrices allows the user to specify different penalties to be placed on the predicted errors resulting in a better tuned controller. The resulting control law that minimizes J is

$$\Delta \mathbf{m} = (\mathbf{A}^T \Gamma_{\boldsymbol{u}} \mathbf{A} + \Gamma_y)^{-1} \mathbf{A}^T \Gamma_{\boldsymbol{u}} \hat{\mathbf{E}}' = \mathbf{K}_c \hat{\mathbf{E}}' \tag{11.44}$$

The weighting matrices $\Gamma_{\boldsymbol{u}}$ and Γ_y contains a potentially large number of design parameters. It is usually sufficient to select $\Gamma_u = \mathbf{I}$ and $\Gamma_y = f\mathbf{I}$ ($\mathbf{I}$ is the identity matrix and f is a scalar design parameter). Large values of f penalize the magnitude of $\Delta \mathrm{m}$ more, thus giving less vigorous control. When $f = 0$, the controller gains are very sensitive to U, largely because of the ill-conditioning of $\mathbf{A}^T \mathbf{A}$, and U must be made small.

11.4 General Guidelines

11.4.1 Effect of Characteristic Process Parameters on Control

The characteristic parameters of a process have a significant effect on how well a controller is able to attenuate disturbances to the process. In many cases, the process itself is able to attenuate disturbances and can be used in conjunction with the controller to achieve better control. The following is a brief discussion outlining the effect of capacity and dead time on the control strategy of a plant.

Capacity

The ability of a system to attenuate incoming disturbances is a function of the capacitance of a system and the period of the disturbances to the system. From **Section 11.2.1 - Terminology**, attenuation is defined as:

$$\text{Attenuation} = 1 - \frac{K}{\sqrt{(\omega\tau)^2 + 1}} \tag{11.45}$$

The time constant, τ, is directly proportional to the capacity of a linear process system. The higher the capacity (time constant) is in a system, the more easily the system can attenuate incoming disturbances since the amplitude ratio decreases. The frequency of incoming disturbances affects the system's ability to attenuate these disturbances. High-frequency disturbances are more easily attenuated than low-frequency disturbances.

> Note: With capacity-dominated processes (with little or no dead time), proportional-only control can achieve much better disturbance rejection. The system itself is able to attenuate disturbances in the frequency range that the controller cannot. High frequency disturbances can be handled by the system. Low frequency disturbances are handled best with the controller.

Dead Time

The dead time has no effect on attenuating disturbances to open loop systems. However, it does have a significant negative effect on controllability. Dead time in a process system reduces the amount of gain the controller can implement before encountering instability. Because the controller is forced to reduce the gain, the process is less able to attenuate disturbances than the same process without dead time. Tight control is possible only if the equivalent dead time in the loop is small compared to the shortest time constant of a disturbance with a significant amplitude.

It is generally more effective to reduce the dead time of a process than increase its capacity. To reduce dead time:

- Relocate sensor and valves in more strategic locations
- Minimize sensor and valve lags (lags in the control loop act like dead time)

To reduce the lag in a system and therefore reduce the effects of dead time, you can also modify the controller to reduce the lead terms to the closed-loop response. This can be achieved by adding derivative action to a controller. Other model-based controller methods anticipate disturbances to the system and reduce the effective lag of the control loop.

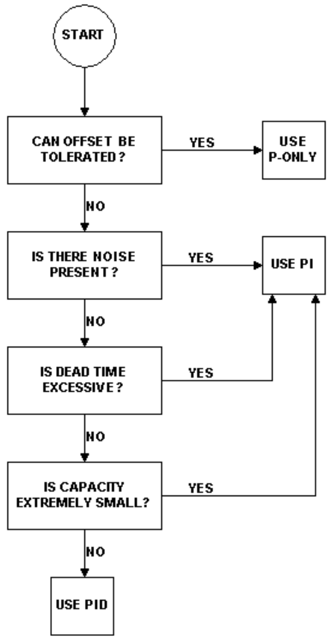
11.4.2 Choosing the Correct Controller

You should consider what type of performance criteria is required for the set point variables, and what acceptable limits they must operate within. Generally, an effective closed loop system, is expected to be stable and cause the process variable to ultimately attain a value equal to the set point. The performance of the controller should be a reasonable compromise between performance and robustness.

A very tightly tuned or aggressive controller gives good performance but is not robust to process changes. It could go unstable if the process changes too much. A very sluggishly-tuned controller delivers poor performance but will be very robust. It is likely to become unstable.

The following is a flowchart that outlines a method for choosing a feedback controller[2].

Figure 11.16



Flowchart for Controller Selection

In general, if offset can be tolerated, a proportional controller should be used. If there is significant noise, or if there is significant dead time and/or a small capacity in the process, the PI controller should be used. If there is no significant noise in the process, and the capacity of the system is large and there is no dead time, a PID controller may be appropriate.

It is apparent why the PI controller is the most common controller found in a plant. There are three possible conditions that a PI controller can handle, whereas the PID controller requires a specific set of conditions in order to be used effectively.

11.4.3 Choosing Controller Tuning Parameters

The following is a list of general tuning parameters appropriate for various processes[3]. The suggested controller settings are optimized for a quarter decay ratio error criterion. Keep in mind that there is no single correct way of tuning a controller. The objective of control is to provide a reasonable compromise between performance and robustness in the closed loop response.

The following rules are approximate. They will provide you very close to tight control. You can adjust the tuning parameters further if the closed loop response is not satisfactory. Tighter control and better performance can be achieved by increasing the gain. Decreasing the controller gain results in a slower but more stable response.

Generally, proportional control can be considered the principal controller. Integral and derivative action should be used to trim the proportional response. Therefore, the controller gain should be tuned first with the integral and derivative actions set to a minimum. If instability occurs, the controller gain should be adjusted first. Adjustments to the controller gain should be made gradually.

Flow Control

Flow in a pipe is typically a very fast responding process. The dead time and capacity associated with a length of pipe is generally very small. It is therefore not unusual for the process to be limited by the final control element (valve) dynamics. You can easily incorporate valve dynamics in the HYSYS model by modifying the valve parameters in the **Actuator** page of the **Dynamics** tab.

Tuning a flow loop for PI control is a relatively easy task. In order for the flow measurement to track the setpoint very closely, the gain, K_c, should be set between 0.4 and 0.65 and the integral time, T_i, should be set between 0.05 and 0.25 minutes. Since the flow control is fast responding, it can be used effectively as the secondary controller in a cascade control structure. The non-linearity in the control loop may cause the control loop to become unstable at different operating conditions. Therefore, the highest process gain should be used to tune the controller. If a stability limit is reached, the gain should be decreased, but the integral action should not. Since flow measurement is naturally noisy, derivative action is not recommended.

Liquid Pressure Control

Like the flow loop process, the liquid pressure loop is typically very fast. The process is essentially identical to the liquid flow process except that liquid pressure instead of flow is controlled using the final control element.

The liquid pressure loop can be tuned for PI and Integral-only control, depending on your performance requirements. Like flow control, the highest process gain should be used to tune the controller. Typically, the process gain for pressure is smaller than the flow process gain. The controller gain, K_c, should be set between 0.5 and 2 and the integral time, T_i, should be set between 0.1 and 0.25 minutes.

Liquid Level Control

Liquid level control is essentially a single dominant capacity without dead time. In some cases, level control is used on processes which are used to attenuate disturbances in the process. In this case, liquid level control is not as important. Such processes can be controlled with a loosely tuned P-only controller. If a liquid level offset cannot be tolerated, PI level controllers should be used.

There is some noise associated with the measurement of level in liquid control. If this noise can be practically minimized, then derivative action can be applied to the controller. It is recommended that K_c be specified as 2 and the bias term, OP_{ss}, be specified as 50% for P-only control. This ensures that the control valve is wide open for a level of 75% and completely shut when the level is 25% for a setpoint level of 50%. If PI control is desired, the liquid level controller is typically set to have a gain, K_c, between 2 and 10. The integral time, T_i, should be set between 1 and 5 minutes.

Common sense dictates that the manipulated variable for level control should be the stream with the most direct impact on the level. For example, in a column with a reflux ratio of 100, there are 101 units of vapour entering the condenser and 100 units of reflux leaving the reflux drum for every unit of distillate leaving. It makes sense that the reflux flow or vapour boilup be used to control the level of the reflux drum. If the distillate flow is used, it would only take a change of slightly more than 1% in the either the reflux or vapour flow to cause the controller to saturate the distillate valve.

Gas Pressure Control

Gas pressure control is similar to the liquid level process in that it is capacity dominated without dead time. Varying the flow into or out of a vessel controls the vessel pressure. Because of the capacitive nature of most vessels, the gas pressure process usually has a small process gain and a slow response. Consequently, a high controller gain can be implemented with little chance of instability.

The pressure loop can easily be tuned for PI control. The controller gain, K_c, should be set between 2 and 10 and the integral time, T_i, should be set between 2 and 10 minutes.

Like liquid level control, it is necessary to determine what affects pressure the most. For instance, on a column with a partial condenser, you can determine whether removing the vapour stream affects pressure more than condensing the reflux. If the column contains noncondensables, these components can affect the pressure considerably. In this situation, the vent flow, however small, should be used for pressure control.

Temperature Control

Temperature dynamic responses are generally slow, so PID control is used. Typically, the controller gain, K_c, should be set between 2 and 10, the integral time, T_i, should set between 2 and 10 minutes, and the derivative time T_d, should be set between 0 and 5 minutes.

Tuning Methods

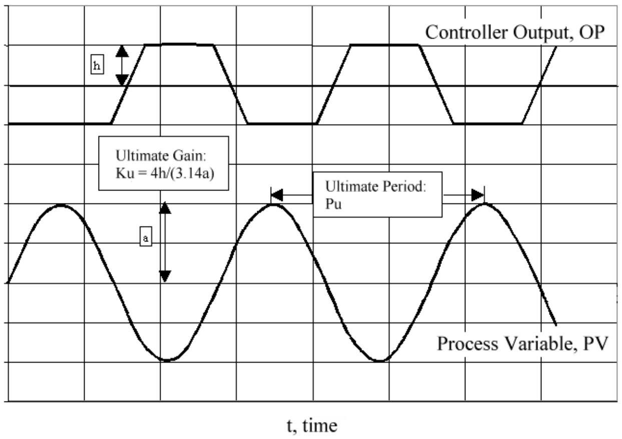
An effective means of determining controller tuning parameters is to bring the closed-loop system to the verge of instability. This is achieved by attaching a P-only controller and increasing the gain such that the closed-loop response cycles with an amplitude that neither falls nor rises over time. At a system's stability margins, there are two important system parameters, the *ultimate period* and the *ultimate gain*, which allow the calculation of the proportional, integral, and derivative gains.

ATV Tuning Technique

The **ATV (Auto Tuning Variation)** technique is used for processes which have significant dead time. A small limit cycle disturbance is set up between the manipulated variable (OP%) and the controlled variable (PV). The ATV tuning method is as follows:

1. Determine a reasonable value for the OP% valve change (h = fractional change in valve position).
2. Move valve +h%.
3. Wait until process variable starts moving, then move valve -2h%.
4. When the process variable (PV) crosses the setpoint, move the valve position +2h%.
5. Continue until a limit cycle is established.
6. Record the amplitude of the response, **A**. Make sure to express **A** as a fraction of the PV span.

Figure 11.17



ATV Stability Limit Parameters

7. The tuning parameters are calculated as follows:

Tuning Parameter	Equation
Ultimate Gain	$K_u = \frac{4h}{\pi a}$
Ultimate Period	P_u = Period taken from limit cycle
Controller Gain	$K_c = \frac{K_u}{3.2}$
Controller Integral Time	$T_i = 2.2P_u$

Ziegler-Nichols Tuning Technique

The **Ziegler-Nichols**[4] tuning method is another method which calculates tuning parameters. The Z-N technique was originally developed for electromechanical system controllers and is based on a more aggressive "quarter amplitude decay" criterion. The Z-N technique can be used on processes without dead time. The procedure as follows:

1. Attach a proportional-only controller (no integral or derivative action)
2. Increase the proportional gain until a limit cycle is established in the process variable, PV.
3. The tuning parameters are calculated as follows:

Tuning Parameter	Equation
Ultimate Gain	K_u = Controller gain that produces limit cycle
Ultimate Period	P_u = Period taken from limit cycle
Controller Gain	$K_c = \frac{K_u}{2.2}$
Controller Integral Time	$T_i = P_u / 1.2$

11.4.4 Basic Steps in setting up a Control Strategy in HYSYS

This section outlines a possible way to create a control strategy in HYSYS. You should first follow the guidelines outlined in **Section 1.4.2 - Moving from Steady State to Dynamics** in order to setup a stable dynamic case. In many cases, an effective control strategy will serve to stabilize the model.

You can install controllers in the simulation case either in **Steady State** or **Dynamic** mode. There are many different ways to setup a control strategy. The following is a brief outline of some of the more essential items that should be considered when setting up controllers in HYSYS.

1. Select the Controlled Variables in the Plant

Plan a control strategy that is able to achieve an overall plant objective and maintain stability within the plant. Either design the controllers in the plant according to your own standards and conventions or model a control strategy from an existing plant. In HYSYS, there are a number a variables which can either be set or controlled manually in a dynamic simulation case. You should distinguish between variables that do not change in a plant and those variables which are controlled.

For more information on setting pressure-flow specifications in a dynamic simulation case, see ***Chapter 1 - Dynamic Theory****.*

Set variables do not change in the dynamic simulation case. Variables such as temperature and composition should be set at each flowsheet boundary feed stream. One pressure-flow specification is usually required for each flowsheet boundary stream in the simulation case. These are the minimum number of variables required by the simulation case for a solution. These specifications should be reserved for variables that physically remain constant in a plant. For example, you can specify the exit pressure of a pressure relief valve since the exit pressure typically remains constant in a plant.

In some instances, you can vary a set variable such as a stream's temperature, composition, pressure or flow. In order to force a specification to behave sinusoidally or ramped, you can attach the variable to the **Transfer Function** operation. A variety of different forcing functions and disturbances can be modelled in this manner.

The behaviour of **controlled variables** are determined by the type of controller and the tuning parameters associated with the controller. Typically, the number of control valves in a plant dictate the possible number of controlled variables. There will be more variables to control in **Dynamic** mode than in **Steady State** mode. For instance, a two-product column in **Steady State** mode requires two **steady state specifications**. The simulator will then manipulate the other variables in the column in order to satisfy the provided specifications and the column material and energy balances. The same column in **Dynamic** mode requires five specifications. The three new specifications correspond to the inventory or **integrating specifications** that were not fixed in steady state. The inventory variables include the condenser level, the reboiler level, and the column pressure.

> A good controller strategy includes the control of both integrating variables and steady-state variables. By maintaining the integrating variables at specified set points, controllers add stability to the plant. Other controllers maintain the desired steady state design specifications such as product composition

2. Select Controller Structures for Each Controlled Variable

You should choose appropriate controller structures for each controlled variable in the simulation case. Some general guidelines in choosing appropriate controllers can be found in **Section 11.4.2 - Choosing the Correct Controller**.

The controller operations can be added in either **Steady State** or **Dynamic** mode. However, controllers have no effect on the simulation in **Steady State** mode. You must specify the following in order to fully define the **PID Controller** operation.

Connections Tab

Process Variable (PV)

The process variable can be specified in the **Connections** tab by pressing the ***Select PV*** button. The controller measures the process variable in an attempt to maintain it at a specified setpoint, SP.

Operating Variable (OP)

The operating variable, OP, can be specified in the **Connections** tab by pressing the ***Select OP*** button. The output of the controller is a control valve. The output signal, OP, is the percent opening of the control valve. The operating variable may be specified as a physical valve in the plant, a material stream, or an energy stream.

Operating Variable	Description
Physical Valve	It is recommended that a physical valve be used as the operating variable for a controller. The controller's output signal, OP, is the desired actuator position of the physical valve. With this setup, a more realistic analysis of the effect of the controller on the process is possible. Material flow through the valve is calculated from the frictional resistance equation of the valve and the surrounding unit operations. Flow reversal conditions are possible and valve dynamics may be modelled if a physical valve is chosen.

Operating Variable	Description
Material Stream	If a material stream is chosen as an operating variable, the material stream's flow becomes a P-F specification in the dynamic simulation case. You must specify the maximum and minimum flow of the material stream by pressing the ***Control Valve*** button. The actual flow of the material stream is calculated from the formula: $Flow = \frac{OP(\%)}{100}(Flow_{max} - Flow_{min}) + Flow_{min}$ HYSYS varies the flow specification of the material stream according to the calculated controller output, OP. (Therefore, a non-realistic situation may arise in the dynamic case since material flow is not dependent on the surrounding conditions.)
Energy Stream	If an energy stream is chosen as an operating variable, you may choose a **Direct Q** or a **Utility Fluid** Duty Source by pressing the ***Control Valve*** button. If the **Direct Q** option is chosen, you must specify the maximum and minimum energy flow of the energy stream. The actual energy flow of the energy stream is calculated similarly to the material flow: $Energy\ Flow = \frac{OP(\%)}{100}(Flow_{max} - Flow_{min}) + Flow_{min}$ If the **Utility Fluid** option is chosen, you need to specify the maximum and minimum flow of the utility fluid. The heat flow is then calculated using the local overall heat transfer coefficient, the inlet fluid conditions, and the process conditions.

Parameters Tab

The direction of the controller, the controller's PV range, and the tuning parameters can be specified in the **Parameters** tab.

A **controller's direction** (whether it is direct or reverse acting) is specified using the **Action** radio buttons. For more information regarding whether a controller is direct or reverse acting, see Section 11.2.1 - Terminology.

A **controller's PV span** is also specified in the PV Range field. A controller's PV span must cover the entire range of the process variable the sensor is to measure.

Tuning parameters are specified in the tuning field. For more information regarding the choice of tuning parameters for each controller, see Section 11.4.3 - Choosing Controller Tuning Parameters**.**

For more information regarding the characterization of final control elements in HYSYS, see ***Modelling Hardware Elements*** *in* ***Section 11.2.2 - Available Control Operations***

3. Final Control Elements

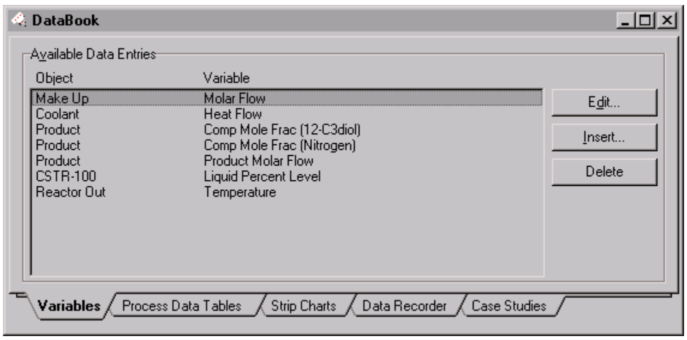
Set the range on the control valve at roughly twice the steady state flow you are controlling. This can be achieved by sizing the valve with a pressure drop between and 15 and 30 kPa with a valve percent opening of 50%. If the controller uses a material or energy stream as an operating variable (OP), the range of the stream's flow can be specified explicitly in the FCV view of the material or energy stream. This view is displayed by clicking on the ***Control Valve*** button in the **PID Controller** view.

The final control element can be characterized as a linear, equal percentage, or quick opening valve. Control valves also have time constants which can be accounted for in HYSYS. It is suggested that a linear valve mode be used to characterize the valve dynamics of final control elements. This causes the actual valve position to move at a constant rate to the desired valve positions much like an actual valve in a plant. Since the actual valve position does not move immediately to the OP% set by the controller, the process is less affected by aggressive controller tuning and may possibly become more stable.

4. Set up the Databook and Strip Charts

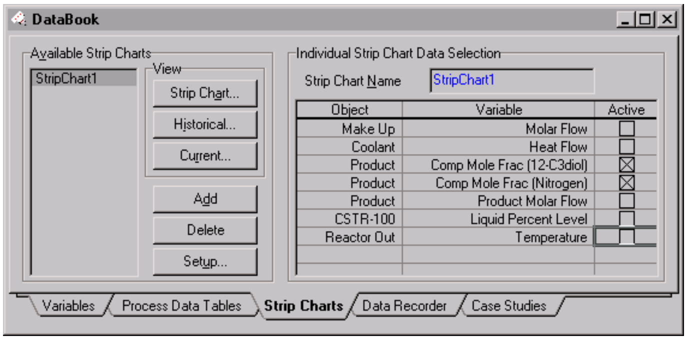
Set up strip charts for your model. Enter the **Databook** property view. Select the desired variables that are to be included in the strip chart in the **Variables** tab.

Figure 11.18



From the **Strip Charts** view, add a new strip chart by pressing the ***Add*** button and activate the variables to be displayed on the strip chart. No more than six variables should be chosen for each strip chart in order to keep it readable.

Figure 11.19



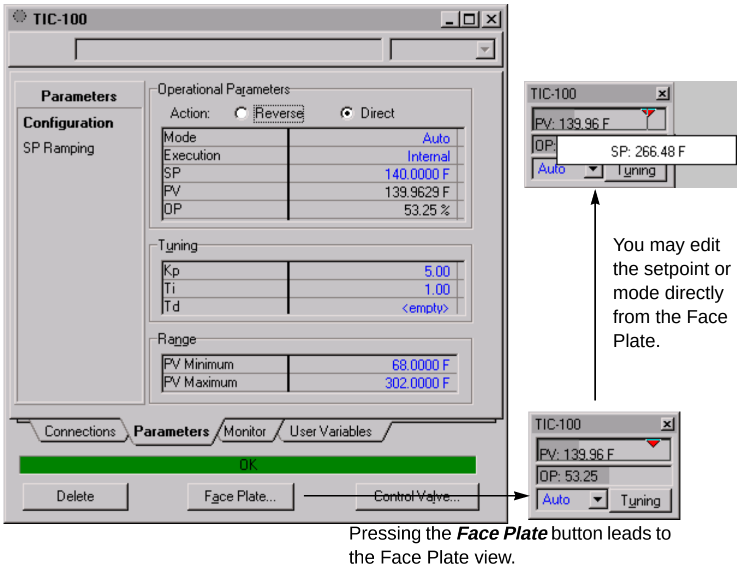
Click on the ***Strip Chart*** button in the **View** group box to see the strip chart. Size as desired and then double click on the strip chart. There are three tabs, where you can set the numerical ranges of the strip chart for each variable, the nature of the lines for each variable, and how the strip chart updates and plots the data.

Add additional strip charts as desired by going back into the **Databook** property view and going to the **Strip Charts** tab.

5. Set up the Controller Faceplates

Click on the ***Faceplate*** button in the **PID Controller** view to display the controller's faceplate. The faceplate displays the PV, SP, OP, and mode of the controller. Controller faceplates can be arranged in the HYSYS work environment to allow for monitoring of key process variables and easy access to tuning parameters.

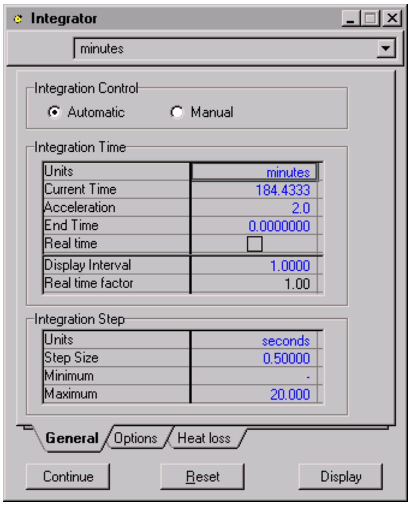
Figure 11.20



6. Set up the Integrator

The integration step size can be modified in the **Integrator** view located in the **Simulation** menu. If desired, change the integration step size to a smaller interval. The default integration time step is 0.5 seconds. Changing the step size will cause the model to run very slowly, but during the initial switch from **Steady State** to **Dynamic** mode, the smaller step sizes will allow the system to initialize better and enable close monitoring of the controllers to ensure that everything was set up properly. A smaller step size also increases the stability of the model since the solver can more closely follow changes occurring in the plant. Increase the integration step size to a reasonable value when the simulation case has achieved some level of stability. Larger step sizes increase the speed of integration and may be specified if the process can maintain stability.

Figure 11.21



7. Fine Tuning of Controllers

Before the Integrator is run, each controller should be turned off and then put back in manual mode. This will initialize the controllers. Placing the controllers in manual will default the setpoint to the current process variable and allow you to “manually” adjust the valve % opening of the operating variable.

If reasonable pressure-flow specifications are set in the dynamic simulation and all the equipment is properly sized, most process variables should line out once the Integrator is run. The transition of most unit operations from **Steady State** to **Dynamic** mode is very smooth. However, controller tuning is critical if the plant simulation is to remain stable. Dynamic columns, for instance, are not open loop stable like many of the unit operations in HYSYS. Any large disturbances in the column may result in simulation instability.

Once the Integrator is running:

1. Slowly bring the controllers on-line starting with the ones attached to upstream unit operations. The control of flow and pressure of upstream unit operations should be handled initially since these variables have a significant effect on the stability of downstream unit operations.

2. Concentrate on controlling variables critical to the stability of the unit operation. Always keep in mind that upstream variables to a unit operation should be stabilized first. For example, the feed flow to a column should be controlled initially. Next, try to control the temperature and pressure profile of the column. Finally, pay attention to the accumulations of the condenser and reboiler and control those variables.
3. Start conservatively using low gains and no integral action. Most unit operations can initially be set to use P-only control. If an offset cannot be tolerated initially, then integral action should be added.
4. Trim the controllers using integral or derivative action until satisfactory closed-loop performance is obtained.
5. At this point, you can concentrate on changing the plant to perform as desired. For example, the control strategy can be modified to maintain a desired product composition. If energy considerations are critical to a plant, different control strategies may be tested to reduce the energy requirements of unit operations.

Stability

It has been shown that the stability of a closed loop process depends on the controller gain. If the controller gain is increased, the closed loop response is more likely to become unstable. The controller gain, Kc, input in the PID Controller operation in HYSYS is a unitless value defined in **Equation (11.46)**.

$$K_c = \frac{OP\% \times PV \text{ Range}}{error} \tag{11.46}$$

In order to control the process, the controller must interact with the actual process. This is achieved by using the **effective gain**, K_{eff}, which is essentially the controller gain with units. The effective gain is defined as:

$$K_{eff} = \frac{K_c(Flow_{max} - Flow_{min})}{PV \text{ Range}} \tag{11.47}$$

Note: *The process gain has units which are reciprocal to the effective gain*

> The stability of the closed-loop response is not only dependent on the controller gain, K_c, but also on the PV range parameters provided and the maximum flow allowed by the control valve. Decreasing the PV range increases the effective gain, K_{eff}, and therefore decreases the stability of the overall closed-loop response. Decreasing the final control element's flow range decreases the effective gain, K_{eff}, and therefore increases the stability of the closed-loop response.

It is therefore possible to achieve tight control in a plant and to have the simulation case become unstable due to modifications in the PV range or C_v values of a final control element.

You should also consider the effect of interactions between the control loops existing in a plant. Interactions between the control loops change the effective gain of each loop. It is possible for a control loop that was tuned independently of the other control loops in the plant to become unstable as soon as it is put into operation with the other loops. It is therefore useful to design feedback control loops which minimize the interactions between the controllers.

11.5 References

[1] Svrcek, Bill. A Real Time Approach to Process Controls First Edition (1997) p.91

[2] Svrcek, Bill. A Real Time Approach to Process Controls First Edition (1997) p.70

[3] Svrcek, Bill. A Real Time Approach to Process Controls First Edition (1997) p.105-123

[4] Ogunnaike, B.A. and W.H. Ray. Process Dynamics, Modelling, and Control Oxford University Press, New York (1994) p. 531

[5] Seborg, D. E., T. F. Edgar and D. A. Mellichamp. Process Dynamics and Control John Wiley & Sons, Toronto (1989) p.649-667

Index

D

E

F

G

H

I

K

L

M

N

O

P

R

S