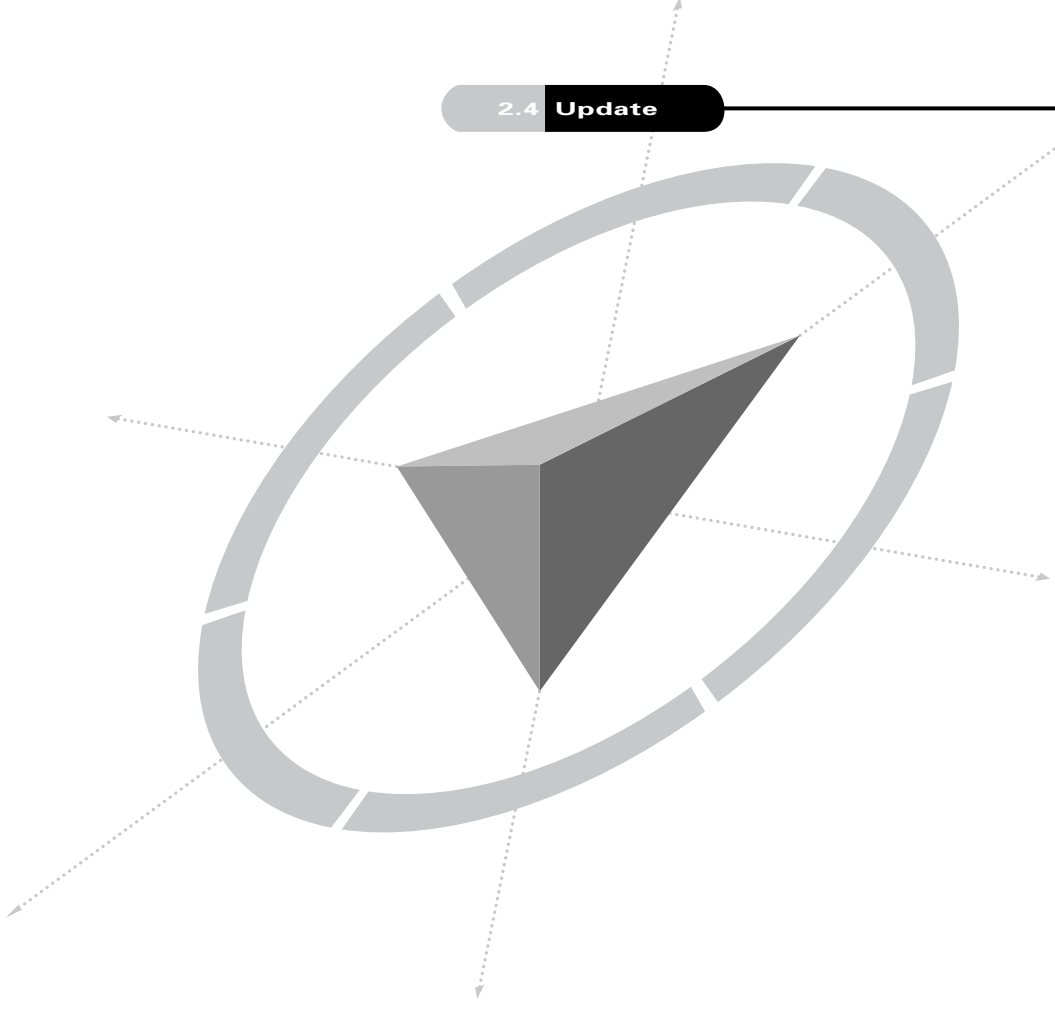
HYSYS

2.4 Update



AEA TECHNOLOGY
THE INNOVATION BUSINESS

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

Copyright Notice

The copyright in this manual and its associated computer program are the property of Hyprotech Ltd. All rights reserved. Both this manual and the computer program have been provided pursuant to a License Agreement containing restrictions on use.

Hyprotech reserves the right to make changes to this manual or its associated computer program without obligation to notify any person or organization. Companies, names and data used in examples herein are fictitious unless otherwise stated.

No part of this manual may be reproduced, transmitted, transcribed, stored in a retrieval system, or translated into any other language, in any form or by any means, electronic, mechanical, magnetic, optical, chemical manual or otherwise, or disclosed to third parties without the prior written consent of Hyprotech Ltd., Suite 800, 707 - 8th Avenue SW, Calgary AB, T2P 1H5, Canada.

© 2001 Hyprotech Ltd. All rights reserved.

HYSYS, HYSYS.Plant, HYSYS.Process, HYSYS.Refinery, HYSYS.Concept, HYSYS.OTS, HYSYS.RTO, DISTIL, HX-NET, HYPROP III and HYSIM are registered trademarks of Hyprotech Ltd.

Microsoft® Windows®, Windows® 95/98, Windows® NT and Windows® 2000 are registered trademarks of the Microsoft Corporation.

This product uses WinWrap® Basic, Copyright 1993-1998, Polar Engineering and Consulting.

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 vii

Hyprotech Software Solutions vii

Use of the Manuals xi

Technical Support xix

1 Interface 1-1

1.1 Interface Basics 1-4

1.2 Manoeuvring Through the Interface 1-20

1.3 Starting a Simulation 1-30

2 Flowsheet Architecture 2-1

2.1 HYSYS Environments 2-4

2.2 Sub-Flowsheet Environment 2-10

2.3 Templates 2-32

3 PFD 3-1

3.1 Object Inspection 3-4

3.2 PFD Tools 3-5

3.3 Installing Streams and Operations 3-9

3.4 Connecting Streams and Operations 3-10

3.5 Disconnecting Streams and Operations 3-13

3.6 The Custom PFD Notebook 3-15

3.7 Cut/Paste Functions 3-17

3.8 Flowsheet Analysis Using the PFD 3-19

3.9 PFD Colour Schemes 3-30

3.10 Manipulating the PFD 3-34

3.11 Adding Annotations 3-52

3.12 Hiding PFD Objects 3-54

3.13 Multi Pane PFDs 3-55

3.14 Printing the PFD 3-57

4 Workbook 4-1

4.1 Opening the Workbook 4-4

4.2 Installing a Stream or Operation 4-5
4.3 Accessing Streams or Operations 4-6
4.4 Deleting Streams or Operations 4-10
4.5 Setup of the Workbook Page 4-11
4.6 Exporting/Importing Workbook Tabs 4-18
4.7 Sorting Information 4-20

5 Flowsheet Analysis in HYSYS 5-1
5.1 Property View Flowsheet Analysis 5-3
5.2 Navigation 5-8
5.3 DataBook 5-15
5.4 Object Status Window/Trace Window 5-46
5.5 Attaching Utilities 5-48

6 Output Control 6-1
6.1 Printing in HYSYS 6-4
6.2 Reports 6-10
6.3 Printing the PFD as a DXF File 6-16
6.4 Graph Control 6-18
6.5 Format Editor 6-23

7 Menu Bar Options 7-1
7.1 File 7-4
7.2 Edit 7-9
7.3 Simulation 7-10
7.4 Flowsheet 7-17
7.5 Workbook 7-25
7.6 PFD 7-26
7.7 Tools 7-27
7.8 Column 7-59
7.9 Basis 7-59
7.10 Window 7-60
7.11 Help 7-62

8 Utilities 8-1
8.1 Boiling Point Curves 8-4
8.2 CO2 Solids 8-10
8.3 Cold Properties 8-12
8.4 Critical Property 8-14
8.5 Data Recon 8-16

8.6 Depressuring 8-17
8.7 Derivative 8-40
8.8 Envelope 8-40
8.9 Hydrate Formation 8-45
8.10 Parametric Utility 8-52
8.11 Pinch Utility 8-65
8.12 Pipe Sizing 8-69
8.13 ..... Property Table 8-73
8.14 Tray Sizing 8-79
8.15 Tray Sizing Manager 8-89
8.16 User Properties 8-109
8.17 Vessel Sizing 8-114
8.18 Adjust Operation with Utility Target Object 8-118
8.19 References 8-120

Index I-1

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

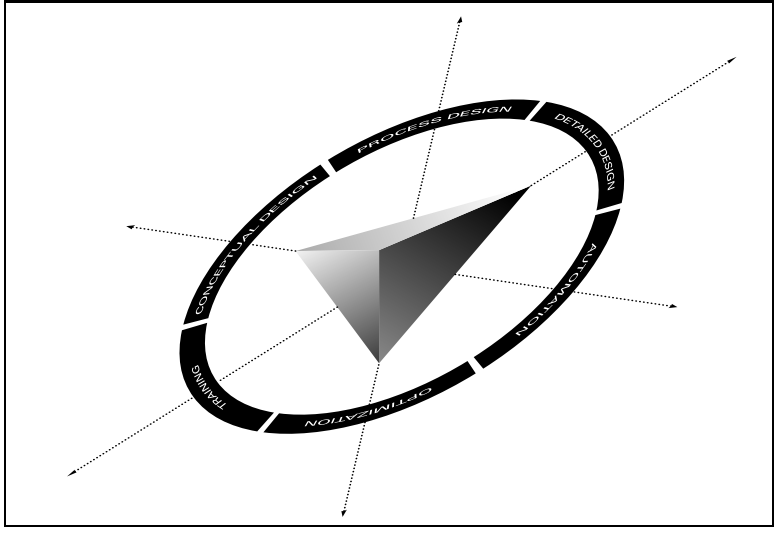
HYSYS has been developed with Hyprotech's overall vision of the ultimate process simulation solution in mind. The vision has led us to create a product that is:

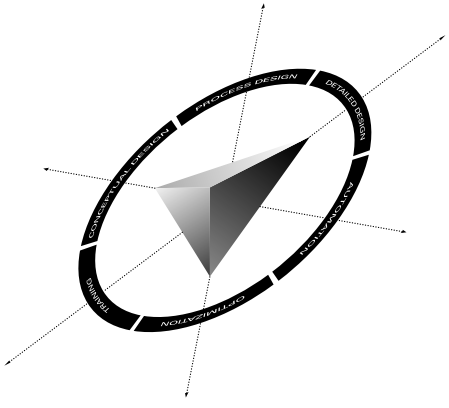
- Integrated
- Intuitive and interactive
- Open and extensible

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.*

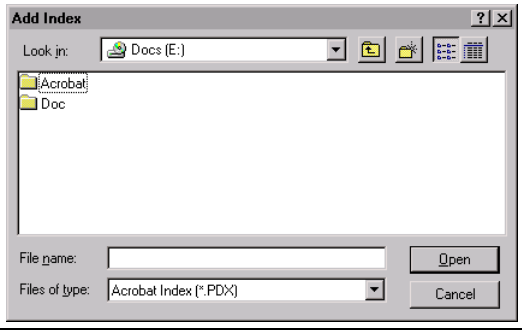
For more information on the search tools available in Acrobat Reader, consult the help files provided with the program.

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 Catalog**®.

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

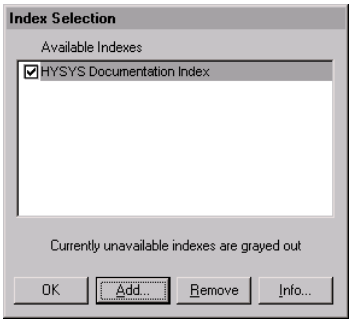
1. Open the **Index Selection** view by selecting **Edit-Search-Select Indexes** from the menu.
2. Click the ***Add*** button. This will open the **Add Index** view.
3. Ensure that the **Look in** field is currently set to your CD-ROM drive label. There should be two directories visible from the root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

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.'

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

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.

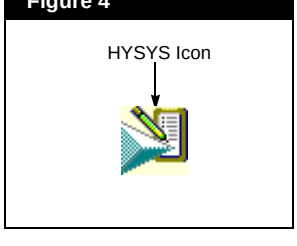
A bulleted list of like objects might describe the various groups on a particular view. For example, the **Options** page of the **Simulation** tab on the Session Preferences view has three groups, namely:

- 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.

Technical Support

There are several ways in which you can contact Technical Support. If you cannot find the answer to your question in the manuals, we encourage you to visit our Website at www.hyprotech.com, where a variety of information is available to you, including:

- answers to frequently asked questions
- example cases and product information
- technical papers
- news bulletins
- hyperlink to support email

You can also access Support directly via email. A listing of Technical Support Centres including the Support email address is at the end of this chapter. When contacting us via email, please include in your message:

- Your full name, company, phone and fax numbers.
- The version of HYSYS you are using (shown in the Help, About HYSYS view).
- The serial number of your HYSYS security key.
- A detailed description of the problem (attach a simulation case if possible).

We also have toll free lines that you may use. When you call, please have the same information available.

Technical Support Centres

Calgary, Canada

AEA Technology - Hyprotech Ltd.
Suite 800, 707 - 8th Avenue SW
Calgary, Alberta
T2P 1H5

support@hyprotech.com (email)
(403) 520-6181 (local - technical support)
1-888-757-7836 (toll free - technical support)
(403) 520-6601 (fax - technical support)
1-800-661-8696 (information & sales)

Barcelona, Spain (Rest of Europe)

AEA Technology - Hyprotech Ltd.
Hyprotech Europe S.L.
Pg. de Gràcia 56, 4th floor
E-08007 Barcelona, Spain

support@hyprotech.com (email)
+34 93 215 68 84 (technical support)
900 161 900 (toll free - technical support - Spain only)
+34 93 215 42 56 (fax - technical support)
+34 93 215 68 84 (information & sales)

Oxford, UK (UK clients only)

AEA Technology Engineering Software
Hyprotech Ltd.
404 Harwell, Didcot
Oxfordshire, OX11 0QJ
United Kingdom

support@hyprotech.com (email)
0800 7317643 (freephone technical support)
+44 1235 434351 (fax - technical support)
+44 1235 435555 (information & sales)

Kuala Lumpur, Malaysia

AEA Technology - Hyprotech Ltd.
Hyprotech Ltd., Malaysia
Lot E-3-3a, Dataran Palma
Jalan Selaman ½, Jalan Ampang
68000 Ampang, Selangor
Malaysia

support@hyprotech.com (email)
+60 3 4270 3880 (technical support)
+60 3 4271 3811 (fax - technical support)
+60 3 4270 3880 (information & sales)

Yokohama, Japan

AEA Technology - Hyprotech Ltd.
AEA Hyprotech KK
Plus Taria Bldg. 6F.
3-1-4, Shin-Yokohama
Kohoku-ku
Yokohama, Japan
222-0033

support@hyprotech.com (email)
81 45 476 5051 (technical support)
81 45 476 5051 (information & sales)

Offices

Calgary, *Canada*
Tel: (403) 520-6000
Fax: (403) 520-6040/60
Toll Free: 1-800-661-8696

Newark, DE, *USA*
Tel: (302) 369-0773
Fax: (302) 369-0877
Toll Free: 1-800-688-3430

Oxford, *UK*
Tel: +44 1235 435555
Fax: +44 1235 434294

Oudenaarde, *Belgium*
Tel: +32 55 310 299
Fax: +32 55 302 030

Hovik, *Norway*
Tel: +47 67 10 6464
Fax: +47 67 10 6465

Kuala Lumpur, *Malaysia*
Tel: +60 3 4270 3880
Fax: +60 3 4270 3811

Yokohama, *Japan*
Tel: 81 45 476 5051
Fax: 81 45 476 3055

Houston, TX, *USA*
Tel: (713) 339-9600
Fax: (713) 339-9601
Toll Free: 1-800-475-0011

Barcelona, *Spain*
Tel: +34 93 215 68 84
Fax: +34 93 215 42 56

Düsseldorf, *Germany*
Tel: +49 211 577933 0
Fax: +49 211 577933 11

Cairo, *Egypt*
Tel: +20 2 7020824
Fax: +20 2 7020289

Seoul, *Korea*
Tel: 82 2 3453 3144 5
Fax: 82 2 3453 9772

Agents

International Innotech, Inc. **Katy,** *USA*	**Tel:** (281) 492-2774 **Fax:** (281) 492-8144
International Innotech, Inc. **Beijing,** *China*	**Tel:** 86 10 6499 3956 **Fax:** 86 10 6499 3957
International Innotech **Taipei,** *Taiwan*	**Tel:** 886 2 809 6704 **Fax:** 886 2 809 3095
KBTECH Ltda. **Bogota,** *Colombia*	**Tel:** 57 1 258 44 50 **Fax:** 57 1 258 44 50
India PVT Ltd. **Pune,** *India*	**Tel:** 91 020 5510141 **Fax:** 91 020 5510069
Logichem Process **Johannesburg,** *South Africa*	**Tel:** 27 11 465 3800 **Fax:** 27 11 465 4548
Plant Solutions Pty. Ltd. **Peregian,** *Australia*	**Tel:** 61 7 544 81 355 **Fax:** 61 7 544 81 644
Protech Engineering **Bratislava,** *Slovak Republic*	**Tel:** +421 7 4488 8286 **Fax:** +421 7 4488 8286
PT. Danan Wingus Sakti **Jakarta,** *Indonesia*	**Tel:** 62 21 567 4573 75/62 21 567 4508 10 **Fax:** 62 21 567 4507/62 21 568 3081
Ranchero Services (Thailand) Co. Ltd. **Bangkok,** *Thailand*	**Tel:** 66 2 381 1020 **Fax:** 66 2 381 1209
S.C. Chempetrol Service srl **Bucharest,** *Romania*	**Tel:** +401 330 0125 **Fax:** +401 311 3463
Soteica De Mexico **Mexico D.F.,** *Mexico*	**Tel:** 52 5 546 5440 **Fax:** 52 5 535 6610
Soteica Do Brasil **Sao Paulo,** *Brazil*	**Tel:** 55 11 533 2381 **Fax:** 55 11 556 10746
Soteica S.R.L. **Buenos Aires,** *Argentina*	**Tel:** 54 11 4555 5703 **Fax:** 54 11 4551 0751
Soteiven C.A. **Caracas,** *Venezuela*	**Tel:** 58 2 264 1873 **Fax:** 58 2 265 9509
ZAO Techneftechim **Moscow,** *Russia*	**Tel:** +7 095 202 4370 **Fax:** +7 095 202 4370

Internet

Website: www.hyprotech.com

Email: info@hyprotech.com

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
Editing/General	CTRL+SHIFT+TAB
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 Interface

1.1 Interface Basics 4

1.1.1 Windows Functionality 4
1.1.2 Primary Interface Elements 4
1.1.3 Multi-Flowsheet Architecture/Environments 5
1.1.4 Objects 6
1.1.5 Structure Terminology 10
1.1.6 Desktop 11
1.1.7 Interface Terminology 13
1.1.8 The Property View 16
1.1.9 Selecting Items 19

1.2 Manoeuvring Through the Interface 20

1.2.1 Hot Keys - alt key 20
1.2.2 Moving Through a View 21
1.2.3 Moving Between Views 22
1.2.4 Supplying Input 24
1.2.5 Editing Input 28
1.2.6 Closing Views 29

1.3 Starting a Simulation 30

1.3.1 File Menu 30
1.3.2 Basis Manager 31
1.3.3 Object Palette 33
1.3.4 Installing Streams 35
1.3.5 Installing Operations 44
1.3.6 Deleting Streams and Operations 51

As you use HYSYS, it will become apparent that you are allowed a high degree of flexibility when performing tasks. In most instances, there is more than one way to accomplish what you are trying to do. Combine this with a consistent and logical approach to how these capabilities are delivered and you are presented with an extremely versatile process simulation tool.

The usability of HYSYS can be attributed to **four key aspects** of its design:

1. First there is the **Event Driven** operation. This concept combines the power of interactive simulation with instantaneous access to information. Interactive simulation means that information is processed as you supply it, with calculations performed automatically. At the same time, you are not tied to the specific location of the program where you are supplying the information. You can access whatever information you need when you need it.
2. Secondly, **Modular Operations** are combined with the **Non-Sequential** solution algorithm. Not only is information processed as you supply it, but the results of any calculation are automatically propagated throughout the Flowsheet, both forwards and backwards. The modular structure of the operations means that they can calculate in either direction, using information in an outlet stream to calculate inlet conditions. You gain process understanding at every step, since operations calculate automatically. You see the results immediately, thus understanding each piece of the simulation in the greatest possible detail.
3. The third attribute is the **Multi-Flowsheet Architecture** which allows you to create any number of Flowsheets within a simulation. In addition to satisfying the technical requirement of using multiple property packages within a simulation, this architecture complements the approach to interactive simulation modeling. You can easily break a large simulation into smaller parts to provide a convenient means of focusing your attention on specific sections of the process.
4. Ultimately, what brings all of these features together is the **Object Oriented Design** of HYSYS. The separation of interface elements (how the information is displayed) from the underlying engineering code means that the same information can be displayed simultaneously in a variety of locations. Each display is tied to the same process variable, so that if the information changes, it is automatically updated in every location. Also, if a variable has been specified, it is shown as a specification in every location - you are not restricted to a single location to change a specification; rather, you can change the specification wherever it is displayed.

1.1 Interface Basics

1.1.1 Windows Functionality

HYSYS has the same basic features as found in other Windows 95 or NT 4.0 based programs.

- ***Minimize***, ***Maximize/Restore*** and ***Close*** buttons located in the upper right corner of most views.
- Close box located in the upper left corner of most views, which contains the normal Windows 3.x menu.

Most of the different views found in HYSYS are resizable to some degree. The following list provides a brief description on resizable views:

- ***Minimize***, ***Maximize/Restore*** and ***Close*** buttons available - view can be resized vertically and horizontally.
- ***Minimize*** and ***Close*** buttons available - view can be resized vertically only.
- ***Close*** button or ***Close*** and ***Pin*** buttons available - view can not be resized.

1.1.2 Primary Interface Elements

Although information can be supplied and accessed a variety of ways, there are four primary interface elements for interacting with HYSYS.

- The PFD is a graphical environment for building your Flowsheet and examining process connectivity. Process information can be displayed for each individual stream or operation as needed.
- The Workbook is a collection of tabs, displaying information in a tabular format. Each Workbook tab displays information about a specific object type, i.e., all streams, pipes, controllers, etc., on a single tab. Multiple tabs for a given object type can be installed, displaying information in varying levels of detail.
- The Property View is a single view which contains multiple tabs. HYSYS extensively uses these single views which include all information about a specific object, i.e. an individual stream or operation.
- The Summary View displays the currently installed streams and operations.

Each of these interface elements, plus the complimentary tools such as the **Data Recorder, Strip Charts, Case Study Tool, Plots**, etc., are all connected through the model itself. A change made in any location is

automatically reflected everywhere else in HYSYS. In addition, there is basically no restriction on what can be displayed at any time. For example, you can have both the PFD and Workbook open, as well as Property Views for operations and streams.

1.1.3 Multi-Flowsheet Architecture/Environments

There are two types of Sub-Flowsheets: Columns and Templates.

As mentioned, HYSYS has been developed around a Multi-Flowsheet Architecture. After you have created the fluid package(s) for your simulation, you enter the Main Flowsheet. In this location, the bulk of the model is created, installing the streams and operations that represent your process.

However, you can create a Sub-Flowsheet at any time. Within the Main Flowsheet, the Sub-Flowsheet appears as a single operation with multiple connections. The Main Simulation has no knowledge of what is inside the Sub-Flowsheet. For instance, it could be a refrigeration loop or a decanter system. From the point of view of the Main Flowsheet, the Sub-Flowsheet simply behaves as any other operation; it calculates whenever conditions are changed.

The nature of the Sub-Flowsheet gives rise to the concept of Environments. Although a Sub-Flowsheet (template or column) appears as a single operation in the Main Flowsheet, you can, at any time, enter the Sub-Flowsheet to examine conditions in greater detail or make changes. When you do this, you enter the Sub-Flowsheet's *Build Environment*. Two things happen at this time. First, the Main Flowsheet is temporarily cached; it will be returned to the exact status when you come back out of the Sub-Flowsheet. Second, the Main Flowsheet solver is temporarily shut down. Calculations will still be performed inside the Sub-Flowsheet, but the results simply won't be propagated to the rest of the simulation until you come back out. This allows you to focus on a specific aspect of the simulation without having the entire simulation calculate every time conditions change.

With this in mind, consider the definition of a Flowsheet (or Sub-Flowsheet) in the context of the overall program. A Flowsheet is defined by what it possesses:

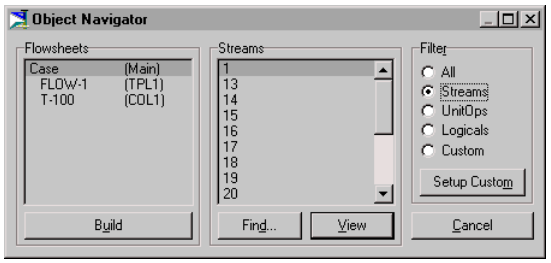
- An independent Fluid Package (optional)
- A PFD
- A Workbook
- Flowsheet Elements (streams and/or operations)

This may seem to contradict what has already been stated regarding access to information. However, capabilities have been built into HYSYS to maximize the power of using Sub-Flowsheets without impeding any access to information. No matter where you are in the simulation, you can open any Flowsheet's PFD or Workbook. Since the Sub-Flowsheets are in essence single operations within the Main Flowsheet, each has its own property view. You can access whatever information resides inside the Sub-Flowsheet through this property view without ever having to enter the Sub-Flowsheet itself.

All of the Flowsheets within a simulation are tied together through the Navigators. The **Object Navigator** allows immediate access to the property view for any stream or operation from any location. Similarly, the **Variable Navigator** allows you to target process variables from any Flowsheet. For instance, you can select variables for inclusion on a Strip Chart or for attachment to logical operations such as Adjusts or Controllers.

*Refer to **Section 5.2 - Navigation** for details on the two navigational tools.*

Figure 1.1



1.1.4 Objects

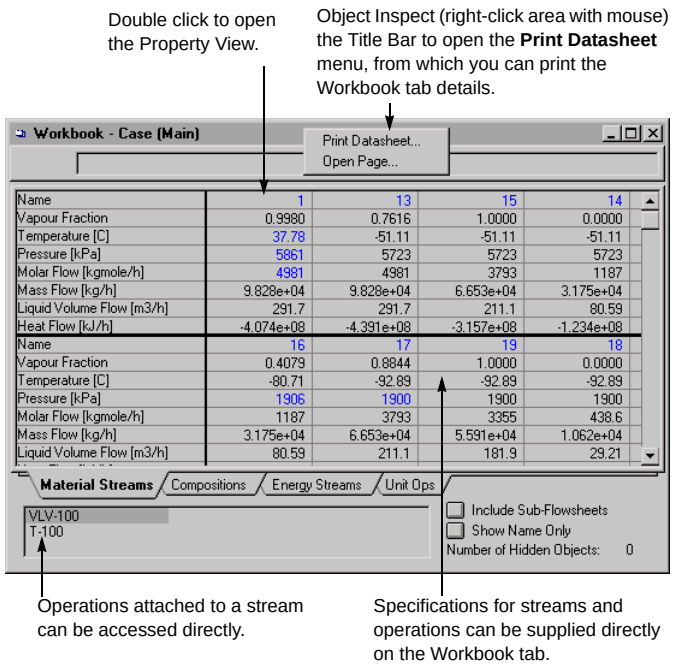
The term Object is used extensively throughout the documentation to refer to an individual stream or operation. Within HYSYS, information associated with an object can be displayed in a variety of ways (Workbook, PFD, Property View, Plot etc.). Through the object oriented design of HYSYS, the information displayed by each interface element is tied to the same underlying object. The result is that if a parameter changes in the Flowsheet it is automatically updated in every location.

Object Inspection is a procedure associated with information display. Object displays (for example an icon in a PFD) are tied to appropriate commands for that object, i.e., printing, direct access to a property view, etc. The way in which the pieces and capabilities of the HYSYS interface work together will be illustrated in the following pages.

On the following pages, the major interface elements of HYSYS, with some of the time saving features built into each, are discussed. Each of these elements can be open simultaneously, and offer different representations of the same information. No matter where you are working - supplying information or installing new streams or operations - every interface element automatically updates with the new information.

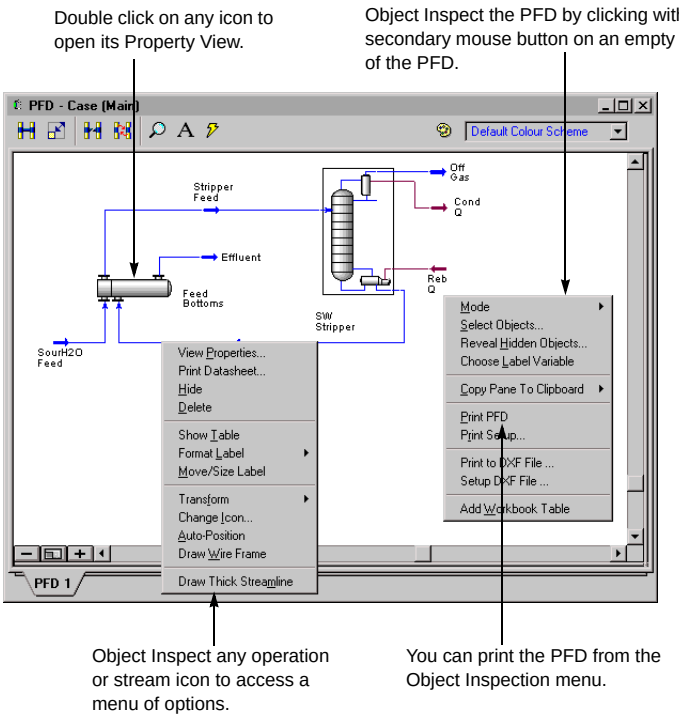
Workbook View

Figure 1.2



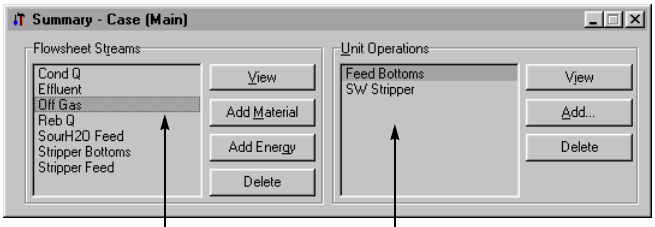
PFD View

Figure 1.3



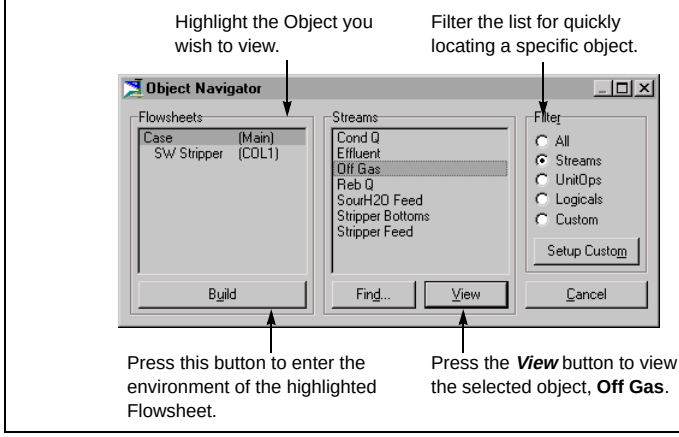
Summary View

Figure 1.4



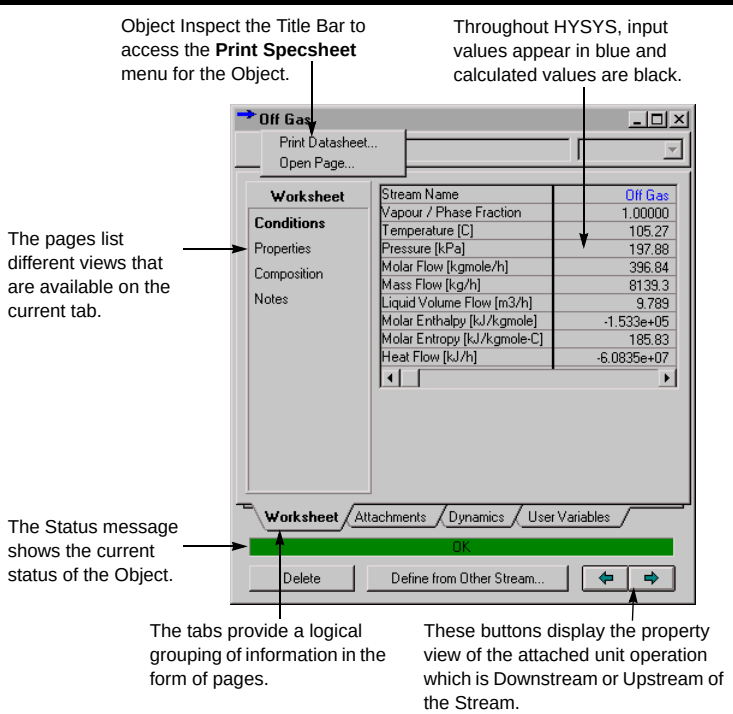
Object Navigator

Figure 1.5



Stream Property View

Figure 1.6



1.1.5 Structure Terminology

Before the HYSYS interface usage is explained, some of the terminology that you will encounter will be defined. Every HYSYS case includes certain structural elements.

Object	Definition
Flowsheet Element (or object)	A Stream or Operation.
Flowsheet	A collection of Flowsheet Elements which utilize a common fluid package. In addition, a Flowsheet possesses its own Workbook and PFD.
Fluid Package	Includes the Property Package, Components (library, pseudo or hypothetical), Reaction Package and User Properties used for Flowsheet calculations. Fluid Packages can be Imported and Exported.
Simulation Case	A collection of Fluid Package(s), Flowsheets, and Flowsheet Elements which form the model. The Simulation Case can be saved to disk for future reference. The extension used for saved cases is hsc.
Session	Encompasses every Simulation Case that you have open while HYSYS is running.

Special Flowsheet Elements

Column Operations and Flowsheet Templates are special Flowsheet Elements in that they are also Flowsheets. A Flowsheet Template can be a Column Sub-Flowsheet or a more complex system.

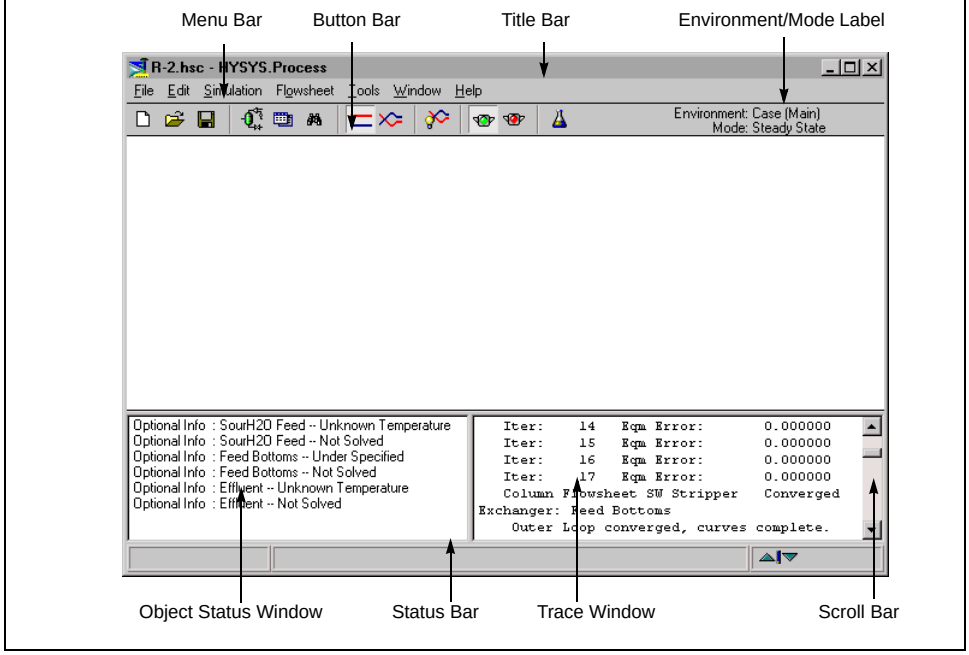
The special capabilities of the Column and Flowsheet Template are:

- They have their own Flowsheet, which means they possess their own Fluid Package, PFD and Workbook.
- They can be comprised of multiple Flowsheet Elements.
- They can be retrieved as a complete entity into any other Simulation Case.

1.1.6 Desktop

The figure below shows the basic components of the HYSYS Desktop:

Figure 1.7



The main features of the Desktop are:

*Refer to the next section, **Button Bar**, for details concerning the various buttons which will be encountered.*

Object	Definition
Title Bar	Indicates the HYSYS file currently loaded.
Menu Bar	Provides access to common Flowsheet commands through a pull down menu system.
Button Bar	Contains various buttons which invoke a specific command when pressed.
Environment/Mode Label	Indicates the environment and mode in which you are currently working.
Status Bar	When the mouse pointer is placed over a button in the Button Bar, the Object Palette, or a Property View, a brief description of its function is displayed in the Status Bar. The Status Bar also displays solver status information.
Calculation/ Responsiveness Button	The Calculation/Responsiveness button allows the user to control how time is spent in updating the screens versus calculations.
Scroll Bar	Scrolls vertically (in the above case) through the Trace Window.

Calculation/Responsiveness Button

For more details on the Object Status Window or Trace Window, refer to ***Section 5.4 - Object Status Window/Trace Window****.*

Object	Definition
Object Status Window/Trace Window	The Object Status Window (left pane) shows current status messages for Flowsheet objects while the Trace Window (right pane) displays Solver information. The windows can be resized vertically or horizontally by clicking and dragging the windows frames located between or above them.

Button Bar

Note that the Button Bar is redrawn depending on the current Environment and Mode.

The buttons on the Button Bar provide immediate access to the most common commands, which are also available as Menu items. The following is a sample of the buttons found on the various Button Bars.

Name	View	Description
New Case		Create a new case.
Open Case		Locate and open an existing case/template/column.
Save Case		Save the active case.
PFD		Open the PFD for the current Flowsheet.
Workbook		Open the Workbook for the current Flowsheet.
Navigator		Open the Object Navigator.
Simulation Navigator		Opens the Simulation Navigator
Dynamics Assistant		Opens the **Dynamic Assistant** view.
Column		Open the **Column Runner** view.
Steady State/ Dynamics		Toggles between **Steady State** and **Dynamic** mode. Currently toggled to Steady State mode.
Active/Holding Run/Stop (Steady State)		Main Environment: Toggles between Active and Holding modes. Green (left) is Active. Column Environment: Toggles between Run and Stop Column Solver. Green (left) is Run.
Integrator (Dynamics)		Integrator toggle. Toggles between Active and Holding. Green (left) is Active.

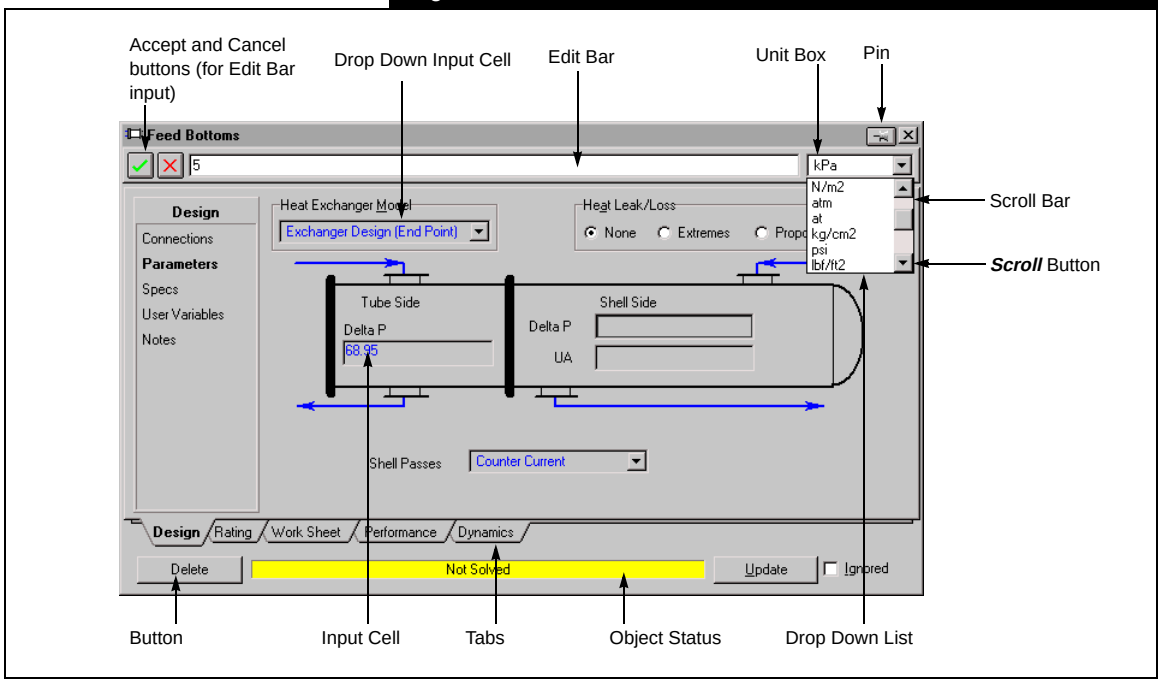
Name	View	Description
Basis		Enter the Basis Environment.
Parent Flowsheet		Return to the parent Flowsheet from a Sub-Flowsheet, i.e., the Main Environment from the Column Sub-Flowsheet Environment.
Oil Environment		Enter the Oil Environment from the Basis Environment.
Leave Environment		From the Oil Environment, return to the Basis Environment; from the Basis Environment, return to the Main Environment.

Some additional things to note about the HYSYS Desktop:

- When the mouse pointer is placed over a button, its descriptive name pops up below the pointer and a flyby function is displayed in the status bar.
- The Desktop itself has both a vertical and horizontal scroll bar. These are automatically created when necessary.

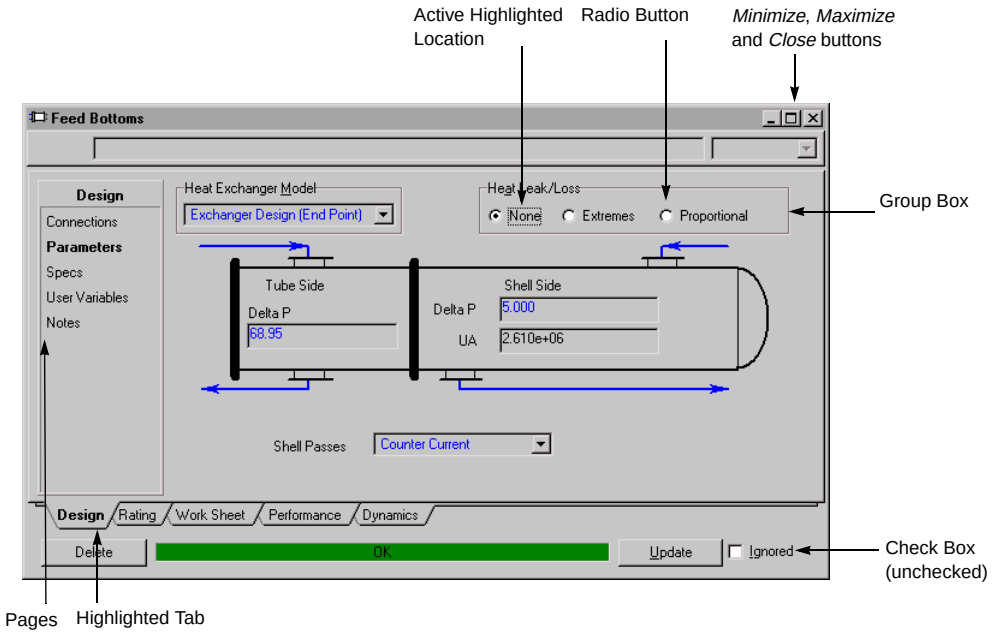
1.1.7 Interface Terminology

Figure 1.8



The following terminology is used to describe the various HYSYS interface elements:

Figure 1.9



Object	Definition
View	Any graphical representation found on the Desktop, for example, a Property View for an operation.
Button	Invokes a command when pressed.
Pin	Converts a Modal Property View to a Non-Modal Property View.
Tabs	Provides a logical grouping of information in the form of pages.
Pages	Provides access to detailed information regarding the selected object.
Drop Down List	A list of available options for a given input cell.
Scroll Bar	Provides access to information which cannot be displayed in the current size of a menu or view.
Scroll Button	Part of the Scroll Bar, allowing you to slide the list up or down, or left or right.
Input Cell	Location in a View for supplying or viewing information, e.g., stream names, temperatures, etc. In many cases it has a drop down list associated with it.

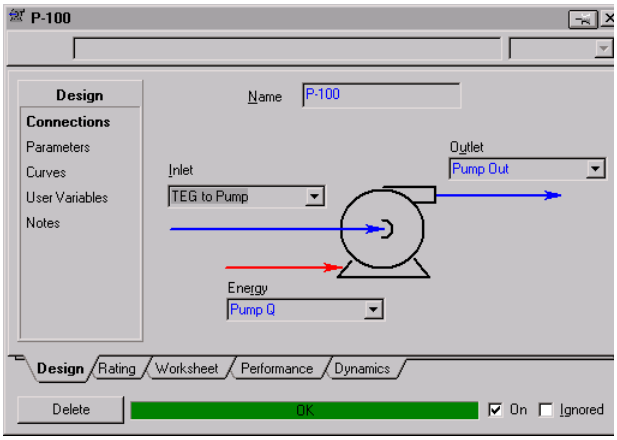
Object	Definition
Edit Bar	Located at the top of a view, and is used for entering or selecting input. The input can be numerical or text.
Unit Box	Associated with the Edit Bar, this provides a drop down list of units which are applicable for the current input cell.
Object Status	Each property view shows the status of the associated object with a coloured background (red for a missing parameter, yellow for a warning message and green for OK).
Icon	A minimized view. Double clicking on one opens the view.
Close Box	Either closes the view (double clicking), or produces a drop down menu of common Windows commands.
Radio Button	Always found in groups of at least two; only one can be active at a time.
Check Box	Items or settings which are On or Off. Selecting the box will turn the function On. Selecting it again turns it Off.
Minimize/Maximize	Either iconize the current view (minimize), or expand a view to its full size.
Active Highlighted Location	The current active location is always indicated by highlighting.
Group Box	Organizational border within a page that groups related functions together. Each group box has its own active location.
Matrix	A group of cells through which you can manoeuvre with the mouse or the keyboard arrow keys.

Active View/Active Location

Although several views can be displayed on the Desktop at any time, only one view is Active or has Focus. This is indicated by the view's Title Bar being highlighted. Within that view, there is again only one location that is Active. Depending on the location (cell, button, etc.), how this is displayed can vary. In the previous view, one example of an Active Location is shown.

1.1.8 The Property View

Figure 1.10



A Property View is a view that contains multiple tabs, contained within a single border. These are used extensively in HYSYS to allow access to all information associated with an item in a single view. A Property View is Modal by default but can be easily converted to a Non-Modal View by selecting the Pin.

A common structure has been adopted for the Property Views. Once you have become familiar with the format and usage of one Property View, each additional one you encounter will behave and be structured in a similar manner.

Accessing Property Pages Via the Page Tabs

Moving between tabs can be done in several ways:

- Select the desired tab with the mouse.
- Use the Hot Key combination **CTRL SHIFT N**, which will cycle you through the tabs (**CTRL SHIFT P** cycles backwards).
- Place the mouse pointer to the direct left or right of the tabs (remaining inside the border of the property view). The cursor will change to a 'tab switch' cursor which points in the direction in which the tabs will be cycled. Click the primary mouse button to cycle through the tabs.

The mouse pointer will not change if the first or last page tab is active AND the pointer is placed to either the left or right, respectively, of the tabs.

Several time saving features have been built into the Property View:

- It will always remember which tab you were last on, returning to that tab automatically the next time you open that view. Each Property View remembers its settings independently; for example, two different streams could automatically open to different tabs.
- There are various ways of moving from one tab to the next including selecting the desired tab or by using the Hot Key combination.

Supplying or specifying input is accomplished in a variety of manners. Some definitions and terminology will be presented in order to adequately explain the functionality and capabilities of HYSYS.

Tabs

Each Property View is made up of four or five tabs, which are displayed at the bottom of the View. These tabs provide a logical grouping for the object information. Note that in Figure 1.11, the **Design** tab is highlighted and appears to be on top of the other tabs, which indicates that this is the current tab.

Pages

In general, each tab will provide access to more than one page or view The section on the left side of the property view has the same name as the current tab. Within this section is a list of available pages, with the current page being highlighted. The information contained on a specific page may be accessed by simply selecting the page from the list. This 'page index' is always present for the specific tab and its appearance is not dependent upon the selection of the page.

Cells

*See **Section 1.2.4 - Supplying Input** for a more detailed description of the Edit Bar.*

This is a location in a view for supplying or viewing information, e.g., stream names, temperatures, etc. Cells can be present as single fields (Input Stream cell on a unit operation) or contained within a group as part of a Matrix. An example of Matrix Cells are those found in the **Worksheet** tab of every unit operation views.

In many cases, cells present as a single field have an associated drop down list, indicated by a down arrow next to the field. This produces a list of available options associated with that field. If the cell is in a Matrix, then the drop down list may be in the Edit Bar.

Highlight Location

The current active location is always indicated by highlighting. Typically this occurs on two levels. First, the actual view (property view for example) will have the Title Bar appear in a different colour than other open inactive views. Second, the active input cell will be indicated with a heavier border, and may also have the contents appear with a highlight. Figure 1.9 shows various types of highlighting. The dashed box in the **Toggles** group is one type and the enlarged size of the page tab for **Simulation** is another highlighting indicator.

Modal versus Non-Modal Property Views

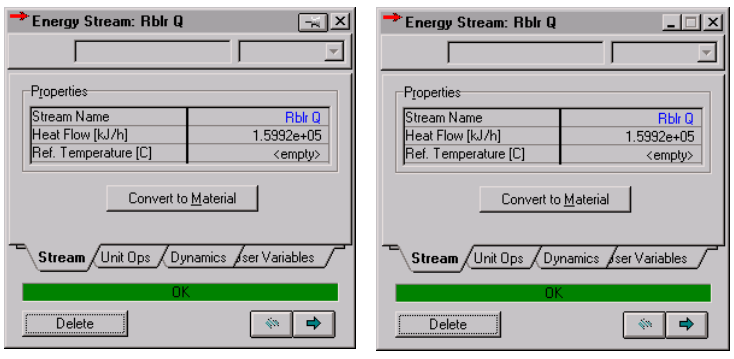
Modal views without pins are called dialog boxes.

When a view is Modal you cannot access any other element in the simulation. That is, you cannot select a menu item or view that is not directly part of that Modal view. This functionality is convenient if you do not want to clutter the Desktop with unneccessary views.

You will encounter Modal Views without Pins, such as the **Input Composition** view, which are called dialog boxes. HYSYS uses dialog boxes to retrieve essential information. In essence, you must either input all required information on this view or input nothing in order to proceed.

Non-Modal Views do not restrict you in this manner. You can leave a Non-Modal View open and interact with any other view or Menu Item by selecting it.

Figure 1.11



Modal View Non-Modal View

You can set Property Views to be Non-Modal by default in the Session Preferences.

The Modal View is indicated by the substitution of the ***Minimize/ Maximize*** buttons with a Pin. The Non-Modal View has the ***Minimize/ Maximize*** buttons in the upper right corner of the window.

A Modal View with a Pin can be converted to a Non-Modal View by selecting that Pin.

1.1.9 Selecting Items

Selecting refers to either picking a specific item from a list, or invoking a button command. There are four types of selecting in HYSYS.

Method	Description
Selecting a cell	Using the mouse, move the cursor to the location and use the primary mouse button to select.
Selecting a button	Select with the primary mouse button, or move the focus location (using the **TAB** key or **SHIFT TAB** combination) and press **ENTER**.
Selecting from a drop down menu or list	Move the focus location into the drop down, and use the mouse pointer or keyboard arrows to highlight the selection, then press **ENTER** to select.
Multiple list selections	Click and drag the mouse over the items you wish to select (provided they are all sequential in the list). Alternatively, you can highlight sequential items by selecting the first item in the list, holding down the **SHIFT** key and selecting the last item in the list. If the items are not sequential, select the first item with the primary mouse button and then select additional items with the mouse while holding down the **CTRL** key.

1.2 Manoeuvring Through the Interface

HYSYS uses Windows conventions regarding interaction between the user and the application. These are described in the following sections.

1.2.1 Hot Keys - ALT KEY

*HYSYS uses a number of Hot Keys for performing commands. For a list, refer to preface of this Manual or to the **Quick Reference Guide.***

Most Labels, including the items on the Menu Bar, input cell identifiers, and Button labels, have a single letter underlined. This letter works in conjunction with the ALT key as a Hot Key combination, the result of which varies depending on the label type.

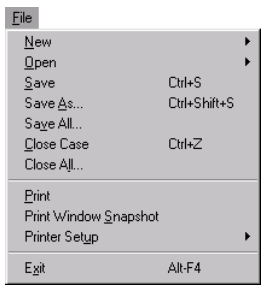
- For Menu Bar Items (**Edit** for example), pressing **ALT E** will open the drop down menu under Edit.
- For Input Cell Identifiers (**Name**, for example), pressing **ALT N** will move the active location directly to that input cell.
- For Buttons (**Close**, for example), pressing **ALT C** will invoke the button command.

The mouse can be used to move the active location to any cell by moving the pointer over the cell and pressing the primary mouse button.

Keep in mind that any keyboard input applies to the active view. For example, if an input cell identifier on the active view has the letter F underlined, you cannot directly open the File Menu by ALT F, as your input will be applied to the active view.

The ALT key by itself automatically advances the active location to the first item in the Menu Bar (**File**). The keyboard arrows move left and right through the row, and the down arrow opens the active menu item. If a drop down menu has underlined letters, you can invoke the command by using that letter only. For example, with the **File** menu open, you can start a New Case by pressing the N key.

Figure 1.12



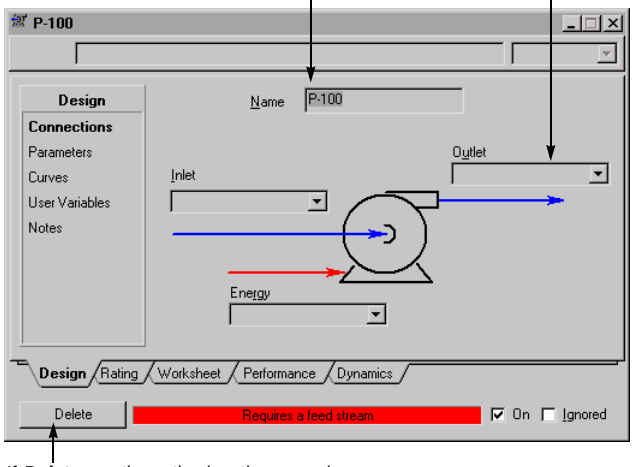
1.2.2 Moving Through a View

When moving through a view using the keyboard functions, you will notice that each input cell and button is accessed sequentially. You can move the active location using the TAB (forward) and SHIFT TAB (reverse) keys. In some instances, such as the Stream property view, HYSYS will automatically advance you to the next input cell after pressing ENTER. If the active location is on a cell in a matrix, the TAB key will not advance you to the next cell in that matrix, but to the next active location in the view. This location may be a button or another group.

HYSYS indicates the active location of the cursor in a view in one of three ways.

- In the case of a string (Stream Name for example), the entire string will be highlighted.
- If the input cell is numerical, a box is placed just inside the cell border.
- In the case of a button, the perimeter of the button will be highlighted and the label surrounded by a dashed box.

Figure 1.13



1.2.3 Moving Between Views

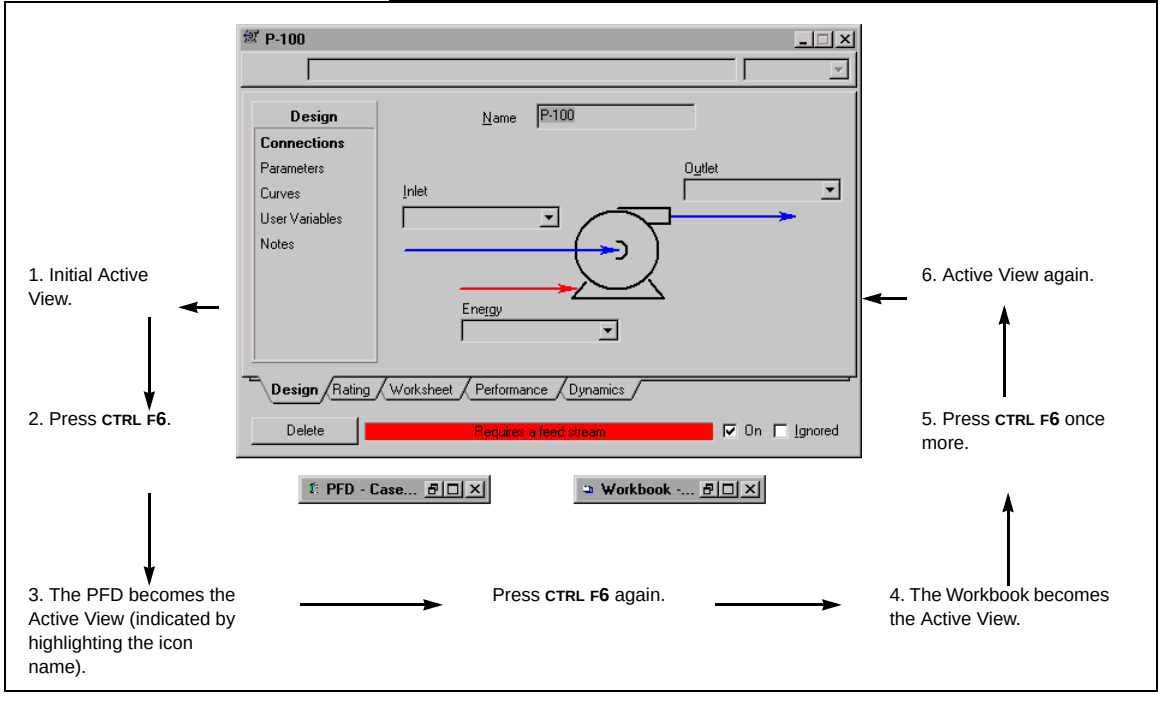
CTRL F6 or CTRL TAB will make the next View in the sequence Active.

CTRL SHIFT F6 or CTRL SHIFT TAB reverses the direction of the sequence.

The easiest way to move from one view to another is to use the mouse. To make a view or icon active, simply press the primary mouse button when the cursor is placed over the view or icon. However, because HYSYS maintains all views (including minimized views) in a sequenced list, you can move to another view using keyboard commands.

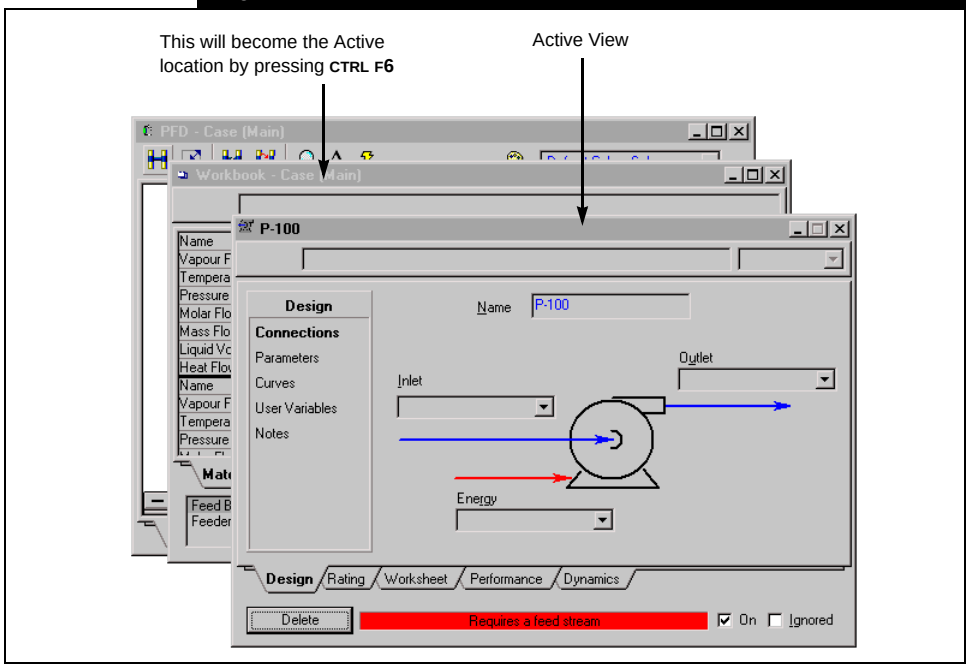
The graphic shown in Figure 1.14 illustrates the method by which you can move between views. If **P-100** is the initial active view, press **CTRL F6**, and the **PFD** icon becomes the active view. This is now highlighted, indicating the active location. Pressing **CTRL F6** again forces the next view in the sequence to become the active location. In this case, the iconized **Workbook**. Press **CTRL F6** once more, and **P-100** will become the active view. In this case, the sequenced views (PFD and Workbook) are minimized. Minimized views which become active are denoted by a highlighted title bar.

Figure 1.14



If all three of the preceding views are open, the view will be similar to that shown below. Sequencing through these will overlay the active view on top of the previously active view (instead of highlighting). In the following example, **P-100** is the active location (notice it is on the top) and its name, **P-100**, is also highlighted. Pressing **CTRL F6** makes **Workbook** the active location. When **Workbook** becomes the active location, **P-100** will be hidden behind the **PFD** view.

Figure 1.15



Moving to the Up/Downstream Object

The stream property view presents a special case of moving to a different view is encountered when it is active. From the stream view, the property views for the upstream and downstream objects can be directly accessed by the ***Upstream Object*** or ***Downstream Object*** button, respectively.

Downstream and Upstream Buttons

1.2.4 Supplying Input

Text Input

When the required input is a **Name** (a stream or operation), you can either supply the input directly via the keyboard, or in most cases choose from a drop down list of applicable responses. If you are supplying the input from the keyboard, (e.g. creating a new stream), simply enter the text and press ENTER.

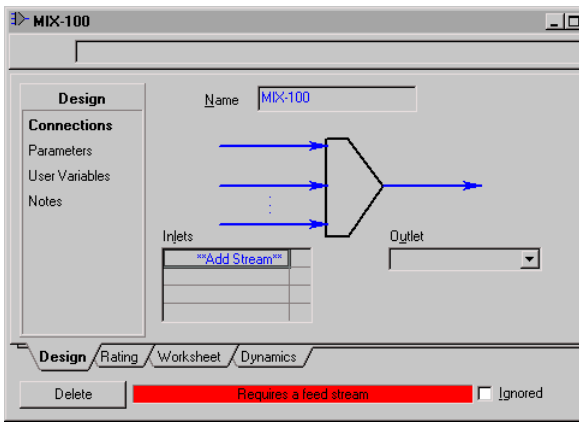
You can override the filtered list by typing in the desired name directly.

The drop down list displays existing objects. By default, HYSYS will filter the list of available streams and operations to include only those applicable to the current situation. For example, when selecting a stream as an operation feed, only unconnected streams or streams that are outlets of other operations will be shown. The following is a description of how to use drop down lists to select existing items.

Depending on the type of cell that has focus, the drop down list may appear next to the cell or in the Edit Bar. A drop down list directly tied to a cell is signified by a down arrow next to the cell (i.e. the **Outlet** cell on the **Mixer** property view).

If a cell's drop down list is in the Edit Bar, then a down arrow will appear in the Edit Bar when the cell has focus. For instance, when a cell in the **Inlets** matrix on the **MIX-100** property view has focus, the Edit Bar contains a drop down list.

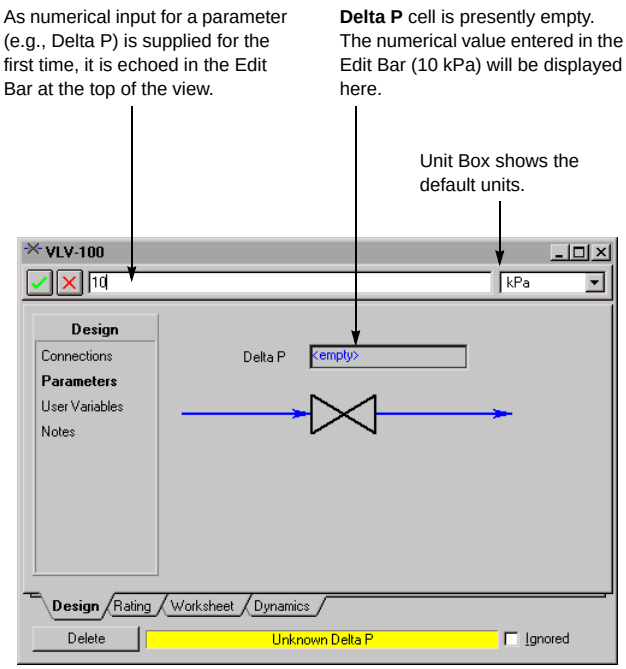
Figure 1.16



Numerical Input

If the input is numerical, the approach is slightly different. Views that accept numerical input use the Edit Bar, which is redrawn to include a Unit Box. The Unit Box will display the current default unit for the input cell property. When you begin supplying a number for a numerical input cell, your input is echoed in the Edit Bar. When you have supplied the number and have pressed ENTER, HYSYS assumes that the default unit was correct and transfers the value back into the input cell.

Figure 1.17

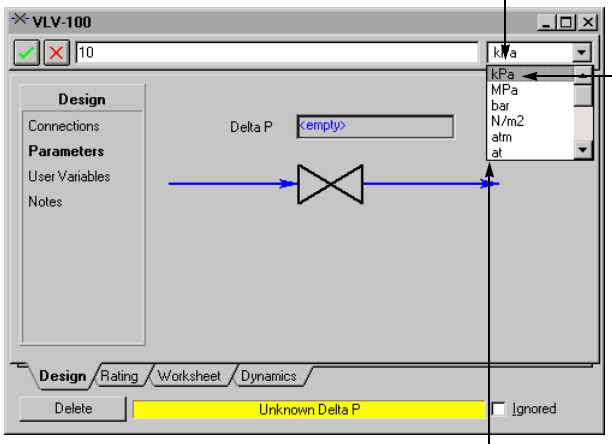


If you are supplying the number with a different unit than the default, there are two methods available for identifying it.

While HYSYS is matching your input to the list of units, you can stop entering text and use any of the other methods for locating the unit (such as selecting it with the mouse, using the up and down arrows etc.).

Method	Action
Keyboard	Input a space after the number and then begin typing in the unit. The unit becomes highlighted in the Unit Box drop down list.
Mouse	After supplying the numerical value, but before pressing ENTER, open the drop down list and locate the desired unit

Figure 1.18



Drop Down Lists and Scroll Bars

Drop down lists are used extensively in HYSYS to provide an efficient means of locating existing streams, operations, or units. These menus can be accessed either via the mouse, or by using keyboard input. Once a drop down list is opened, you manoeuvre through the list using the mouse or keyboard.

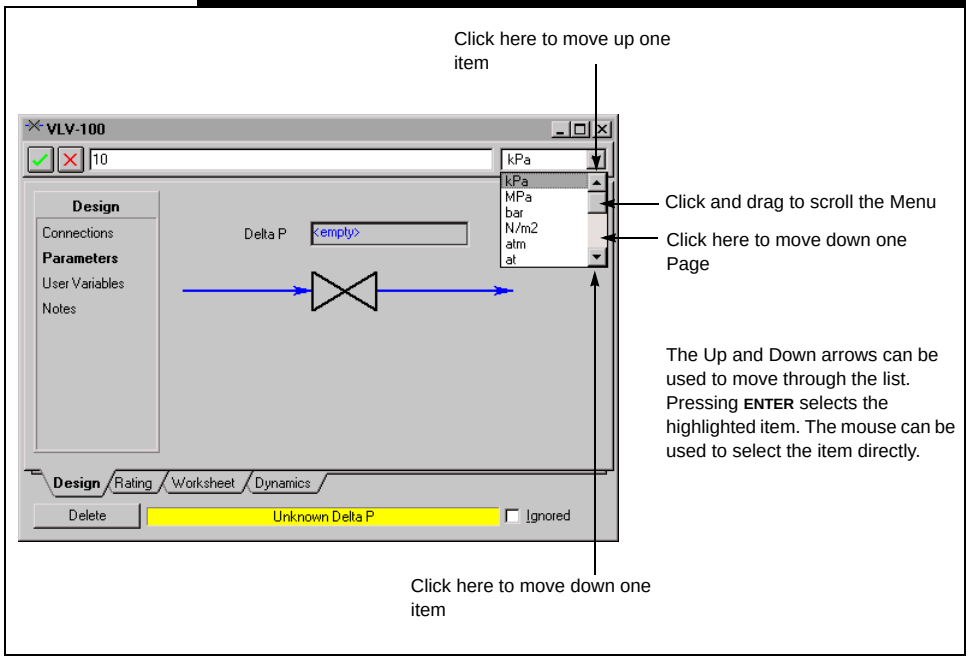
The F2 key opens the drop down list in the Edit Bar.

A drop down list for a Text input cell (valve inlet for example), can be opened at any time by using the primary mouse button to select the applicable arrow box. This not only opens the menu, but moves the active location to that input cell. You can also open the drop down list for the current active cell by pressing the keyboard down arrow. If the stream is being attached for the first time, the highlight in the drop down list will be at the top of the list. If the input cell already has a stream in it, the highlight will be on that stream.

For a numerical input cell, the drop down list is located next to the Edit Bar at the top of the view, and contains the Unit Box. This automatically

opens as soon as you stop supplying numerical input (signified by a **<space>**) and begin entering the unit. Alternatively, the drop down list can be opened at any time (which terminates the number entry) by selecting the arrow box.

Figure 1.19



Once a drop down list is opened, you can manoeuvre through it in several ways:

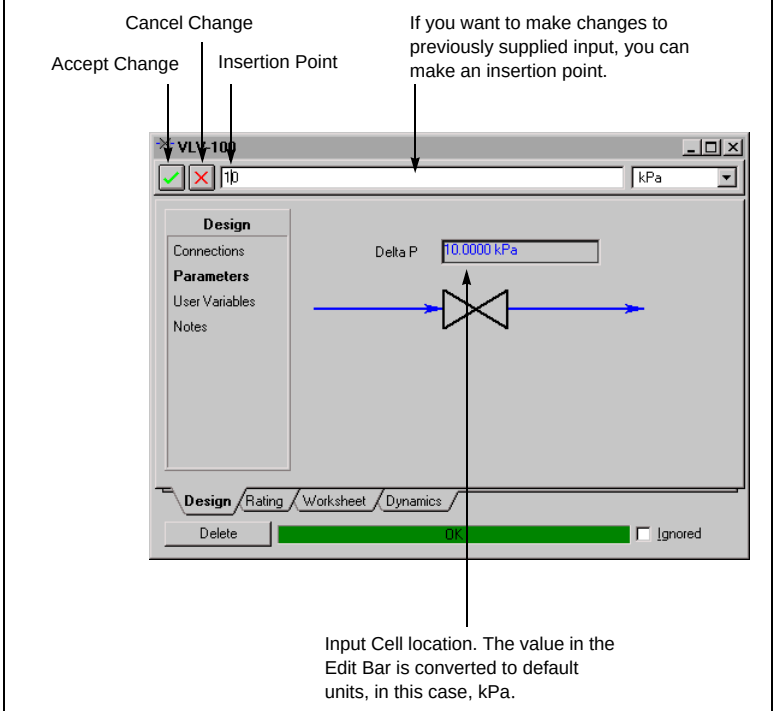
- The most convenient method is via *type-matching*. Once a drop down list is open, keyboard input is interpreted to find the first menu item which best matches your input. As you continue to supply input, the matching continues. Pressing **ENTER** terminates the string and accepts the highlighted item. You can also use the keyboard arrow keys to move to any item.
- If the menu contains six items or less, it appears without a scroll bar. In this case, you can use the mouse to directly select the desired item, or use the up and down keyboard arrows to mark the item and then select it with the **ENTER** key.
- The **PAGE UP** and **PAGE DOWN** keys move the menu by one page, and the **HOME** and **END** keys take you to the first and last item respectively. The desired item is selected by highlighting it and pressing **ENTER**.

- The Scroll Bar/Scroll Button provides similar functionality. Selecting the Up and Down Scroll Arrows advance the menu by one item. The scroll button can be selected with the primary mouse button and dragged up and down to quickly scroll the menu.
- Selecting the space between the scroll button and the scroll arrow advances the menu up or down one page. The desired item can then be selected with the primary mouse button.

1.2.5 Editing Input

Editing input can be done in two ways. When the input cell is active, any information you supply will overwrite the previous input. Likewise, you can use the drop down list to replace previous input.

Figure 1.20



To make an Insertion Point somewhere in the number, use the mouse or press F2 and use the arrow keys. You can then make selective changes.

You have a choice when editing numerical input.

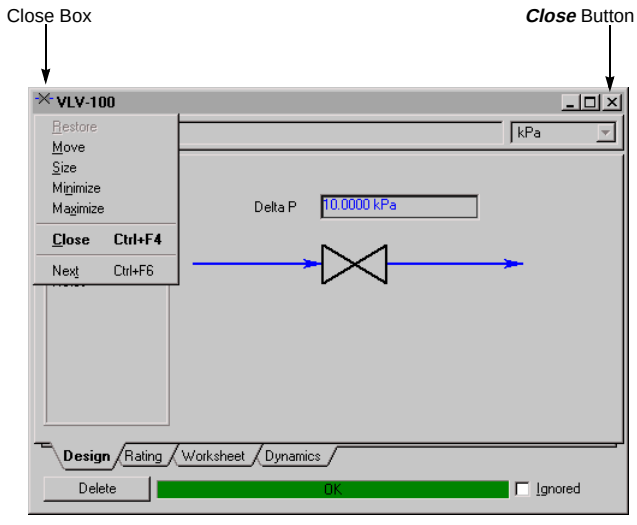
- If you simply type in a new value and press **ENTER**, HYSYS accepts the input and assumes it is in the default units.
- You can change the units using the methods described previously.
- Another method is via selective modification. With this route, you place an insertion point somewhere in the string and make selective changes. Pressing **ENTER** or selecting the **Accept Change** box accepts the changes, while you can abort them by selecting the **Cancel Change** box.

1.2.6 Closing Views

There are four ways to close a view.

- Double click on the **Close Box** in the upper left hand corner of the view.
- Click once on the Close Box to open the drop down menu. The Close function in this drop down menu has the **C** underlined. Press the **C** on the keyboard to close the view.
- **CTRL F4** closes the active view.
- Click on the ***Close*** button in the upper right hand corner of the view.

Figure 1.21



1.3 Starting a Simulation

When starting a New simulation, the first thing you need to do is create a New Case. You can do this through the **File** option in the Menu Bar or using the ***New Case*** button.

New Case Button

1.3.1 File Menu

Before proceeding with the installation of a new case, the options available under **File** in the Menu Bar will be explained. These options are the general commands associated with any file - saving, printing, etc.

Command	Definition
Starting (New/Open)	Creates a New HYSYS case or Opens an existing one.
Saving (Save/Save As/Save All)	When saving a case for the first time, choose the **Save** command. Supply the file name and the location (file path) to which you want to save the file. If the case has been previously saved, this command updates the information on disk. If you want to change the file name or location, use the **Save As** command. The **Save All** command can be used to save all HYSYS cases currently open. You are asked to select which cases should be saved.
Closing (Close Case/ Close All)	**Close Case** closes the current simulation; you are first prompted to save it. **Close All** accesses a dialog of cases currently in memory, allowing you to specify which should be closed. Again, you will be prompted to save the cases first.
Printing (Print/Print Snapshot/Printer Setup)	**Print** accesses a dialog of options for printing Specsheets for the active object. **Print Snapshot** prints a snapshot of what currently appears in the HYSYS active window. **Printer Setup** is used to select the default printer, print orientation, paper size, etc. It is similar to the Printer Setup commands in other Windows applications.

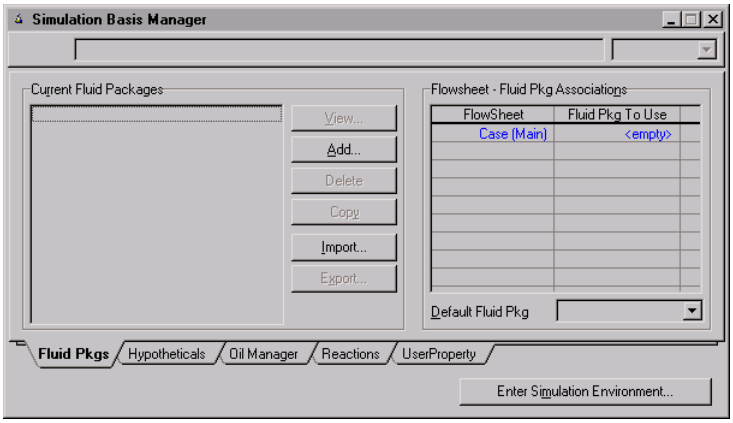
For more information about the File Menu, refer to ***Section 7.1 - File****.*

See ***Chapter 6 - Output Control*** *for more information on printing.*

1.3.2 Basis Manager

When you start a New Case, HYSYS places you directly into the **Simulation Basis Manager**. From this location you can manipulate every Fluid Package in the simulation. The minimum steps required to create a Fluid Package for your simulation are described.

Figure 1.22

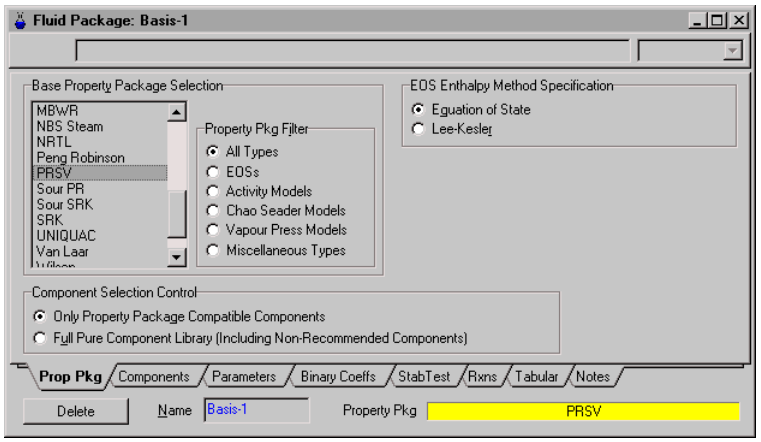


*Refer to **Chapter 1 - Fluid Package** of the **Simulation Basis Manual** for more information.*

From the Basis Manager, select the ***Add*** button to add the Fluid Package to the new case. Prior to entering the Main Environment, you must fully define at least one fluid package by selecting a property package and a set of components. HYSYS places you on the **Prop Pkg** page of the Fluid Package.

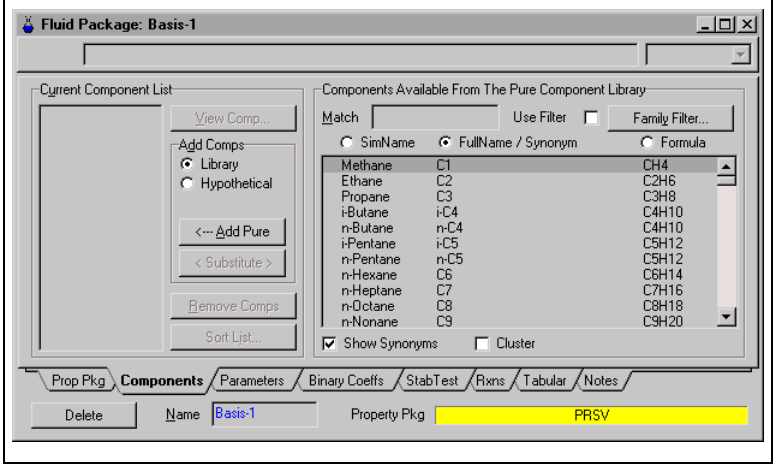
From the **Base Property Package Selection** group, select a Property Package to use with this case.

Figure 1.23



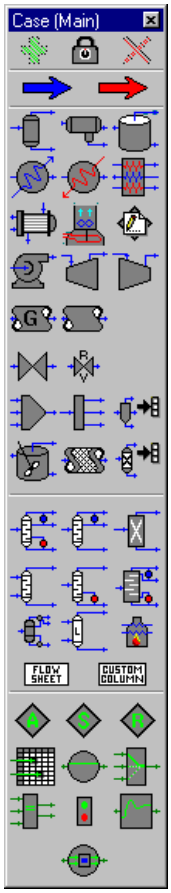
The next thing to do is select the components you are going to use in the case. Select the **Components** tab.

Figure 1.24



Once the Components and the Property Package have been selected, press the ***Close*** button to return to the Basis Manager. From here, press the ***Enter Simulation Environment*** button to move to the Main Environment.

Now that you have entered the Main Environment, you need to install streams and operations in the case.



Object Palette

1.3.3 Object Palette

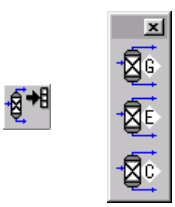
The Object Palette can be used to install streams and operations. You can open or close the Palette from the **Flowsheet** option in the Menu Bar, or by using the **F4** Hot Key.

In the Main Flowsheet or Template Sub-Flowsheet, every operation available in HYSYS is accessible via the Palette, (except those specifically associated with Columns such as tray sections, reboilers, etc.). A separate Palette is produced when you are inside the Column Sub-Flowsheet.

From top to bottom, the Palette is organized into the following categories:

- **Streams**
- **Vessels (2 and 3-phase separators, tank)**
- **Heat Transfer Equipment**
- **Rotating Equipment (compressor, expander, pump)**
- **Piping Equipment**
- **Solids Handling**
- **Reactors**
- **Prebuilt Columns**
- **Shortcut Columns**
- **Sub-Flowsheets**
- **Logicals**

Buttons on the object palette that display an arrow pointing to the side represent general buttons. For instance, the Solid Ops and General Reactors buttons each will bring up a secondary palette displaying buttons for more specific unit operations. As shown, you can select a Gibbs Reactor, an Equilibrium Reactor or a Conversion Reactor after pressing the General Reactors button.



General Reactors

Each operation has a representative icon. In addition, when you place the mouse pointer over top of any button, the flyby description of what operation the icon represents is displayed both below the pointer and at the bottom of the HYSYS Window in the Status Bar.

Installing Streams and Operations via the Palette

Add Button

You can install a single stream or operation from the Palette by double clicking on the icon for the object you want to install. You can also use the ***Add*** button at the top of the Palette. First, click on the button for the Object you wish to install, then click on the ***Add*** button. This displays the Property View for the Object type chosen.

You can install multiple Streams and Unit Operations from the Palette by using the ***Lock*** button. The procedure for this is as follows:

Lock Button

1. Click on the ***Lock*** button at the top of the Palette.
2. With the primary mouse button, click on the Icon for the Stream or Operation you want to install.
3. Select the ***Add*** button to install the stream or operation.
4. You can repeat this as many times as necessary.

Cancel Button

To deactivate the chosen Object button when in locked mode, press the ***Cancel*** button or select a different Object. The Object button returns to its normal state (changes from light grey to dark grey).

To remove the Lock function, click on the ***Lock*** button again.

1.3.4 Installing Streams

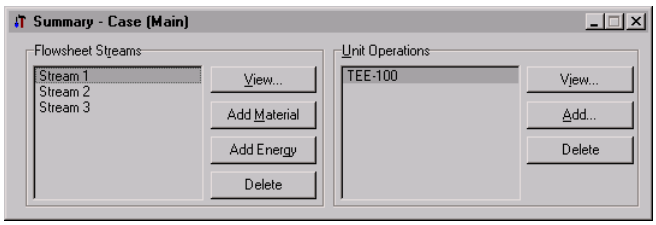
There are several methods to install new streams in HYSYS. Some of these methods immediately open the Stream Property View upon installation and others do not.

Installation with Direct Access to Property Views

*Note that both the menu bar and the F11 hot key options will install a material stream. To convert the material stream to an energy stream, refer to Section 2.1 - Material Stream Property View of the **Steady State Modeling Guide**.*

Method	Description
Menu Bar	New streams can be installed into the case via the Menu Bar. Choose the **Flowsheet** option from the Menu Bar, and then choose the **Add Stream** option.
Hot Keys	Press **F11**.
Object Palette	Double click on the stream icon in the Object Palette.
Summary View	Open the **Select Summary** view by selecting **Summaries** from the **Tools** option in the Menu Bar. From this view, highlight the Flowsheet to which you will be adding the stream operation and press the ***View*** button. The **Summary** view will appear, from which you can choose the ***Add Material*** or ***Add Energy*** buttons in the Flowsheet Streams group.

Figure 1.25



Installation Without Direct Access to Property Views

Method	Description
Workbook	Position the cursor in the ****New**** cell of a *Streams Type* Workbook tab. Type in the new stream name. You can now proceed to enter values for the default stream variables.
Operation Property View	You can supply a new stream name in an Input Cell for an operation, which automatically creates the stream.
Object Palette to PFD	Both Material and Energy Streams can be added to the case via the Object Palette. • With the primary mouse button, click on the Stream Icon in the palette (Material or Energy). Move the cursor to the area on the PFD where the Stream is to be placed. Press the primary mouse button to complete the installation. or • From the Object Palette, click (with the secondary mouse button) and drag the stream icon to the desired location in the PFD. Release the secondary mouse button to drop the stream on the PFD.

Material Stream Button (Blue)

Energy Stream Button (Red)

To access the Property View for a stream, do one of the following:

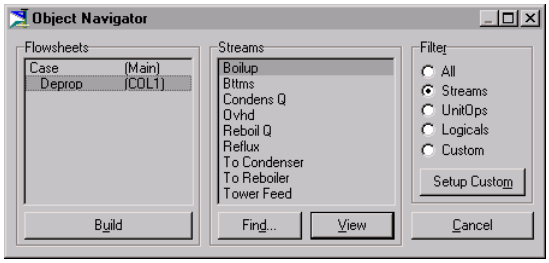
- Double click on a Stream cell (any cell except a flow cell, which accesses the Input Composition dialog) in the Workbook.
- Double click on the Icon in the PFD.
- Object inspect (with the secondary mouse button) a stream cell in the Workbook. Choose View from the menu that appears.
- Object inspect the Icon in the PFD and choose View Properties.
- Open the Object Status Window (left pane). Double click on the message related to the stream.
- Access the Summary view, highlight the stream name in the Flowsheet Streams group and press the ***View*** button.

*Refer to **Section 3.8 - Flowsheet Analysis Using the PFD** for more information on accessing Property Views from the PFD.*

*Refer to **Section 5.2.1 - Object Navigator** for information on accessing the navigator and its functions.*

- Access the Object Navigator. Select the Streams radio button and highlight the name of a stream in the Streams group. Press the ***View*** button.

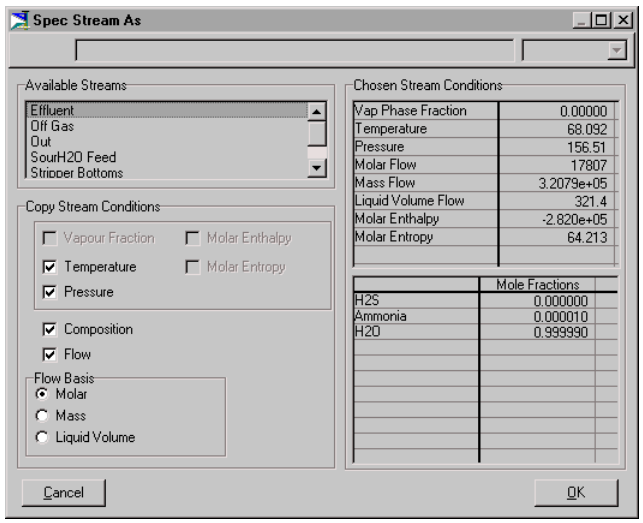
Figure 1.26



Spec Stream As

On each Stream Property View, there is a ***Define from other Stream*** button. This allows you to copy the existing specifications from another stream into the present stream. When you press this button, the **Spec Stream As** view appears.

Figure 1.27



In the **Copy Stream Conditions** group, you can select a maximum of two variables in the first group: vapour fraction, temperature, pressure,

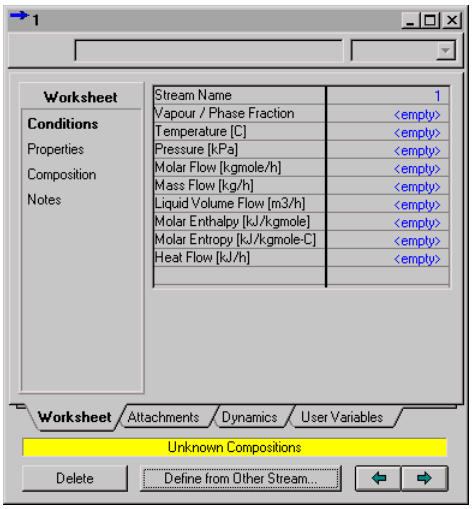
molar enthalpy, or molar entropy. These will become the user defined variables for the stream. If you wish to have the composition and/or flow copied, select the appropriate box. From the **Flow Basis** group, you can select which flow type will be the user defined flow.

Installing a Stream - Example

Material and Energy streams can be installed with any one of the methods mentioned previously. For this example, the default units are SI, the property package is NRTL and the components are Water and Methanol. The following procedure will outline the installation of a Stream:

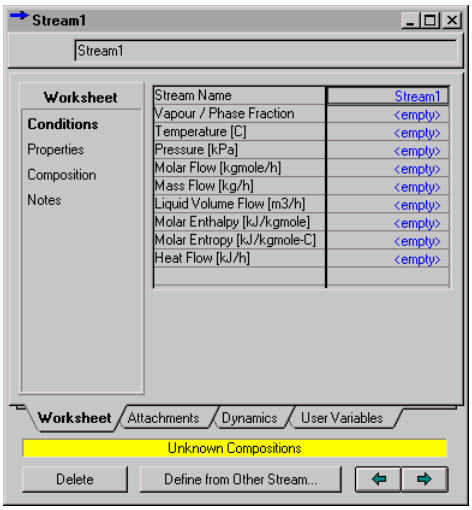
1. Double click on the **Material Stream** button in the Object Palette. This brings up the Stream property view, which will have been auto-named as **1**.

Figure 1.28



2. The **Stream Name** cell will have focus. You can type in a new name, for example, **Stream1**. Your input will be echoed in the Edit Bar. Pressing ENTER transfers your input back to the **Stream Name** cell.

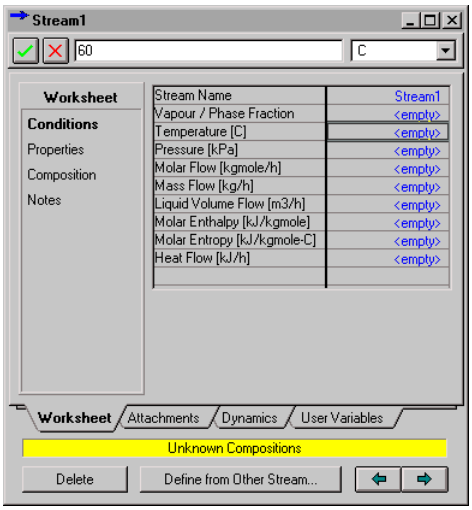
Figure 1.29



3. Nothing will be supplied for **Vapour/Phase Fraction**. Press the keyboard down arrow to move the highlight to the **Temperature** cell.

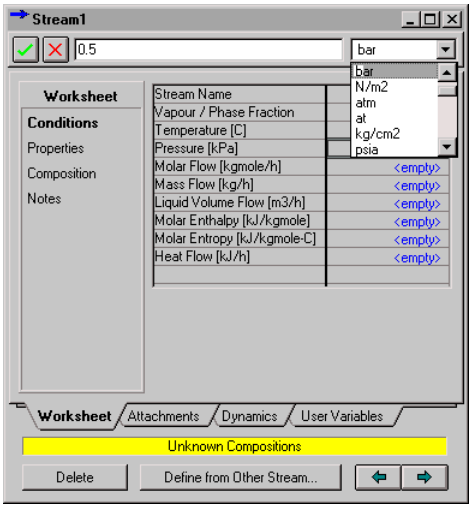
4. With the highlight on the **Temperature** cell, type in **60**. This appears in the Edit Bar. In the Unit List, **C** appears because this is the default SI temperature unit. Press ENTER to accept the entry.

Figure 1.30



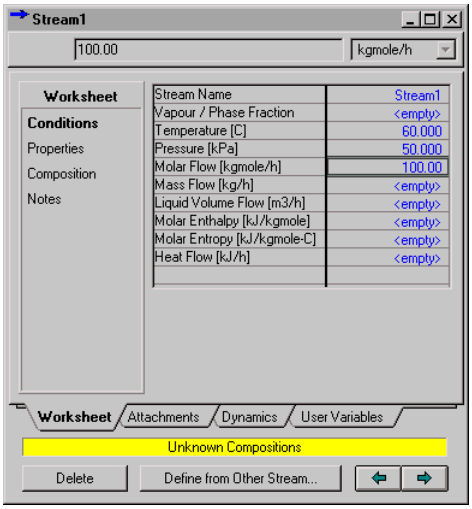
5. The highlight will now be on the **Pressure** cell. You want the pressure for this stream to be 0.5 bar. The default SI unit for pressure is kPa, so you must change the pressure unit to **bar**. Type in **0.5** followed by a Space, and the letter **b**. The Unit Box drop down list will appear and **bar** will be highlighted. Press ENTER to accept the units.

Figure 1.31



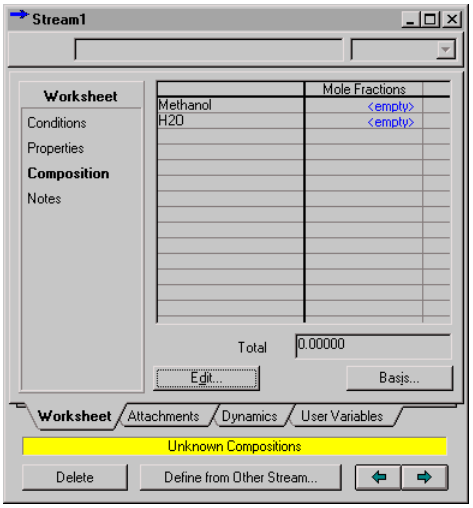
6. Notice that the value transferred into the **Pressure** cell is 50 kPa, as kPa is the default display unit for pressure.
7. You will now be on the **Molar Flow** cell. Type in a value of **100 kgmole/hr** and press ENTER. Notice that the stream still hasn't flashed, because we have not supplied a composition.

Figure 1.32



8. With the primary mouse button, select the **Composition** page. To supply or edit the composition, either select a cell and enter a value or press the ***Edit*** button. The **Input Composition** view will be displayed.

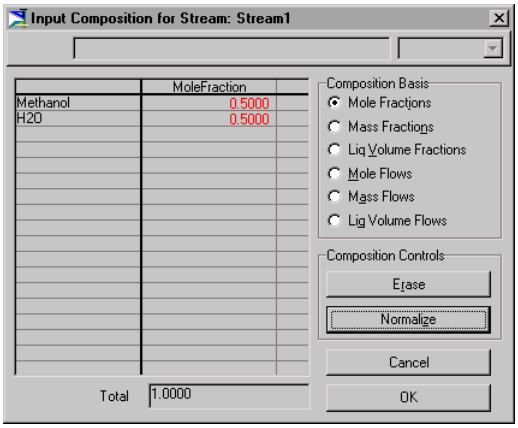
Figure 1.33



Note that you could also double click on the appropriate flow cell (mole, mass, or liquid volume) on the **Conditions** page and access the following **Input Composition** view directly.

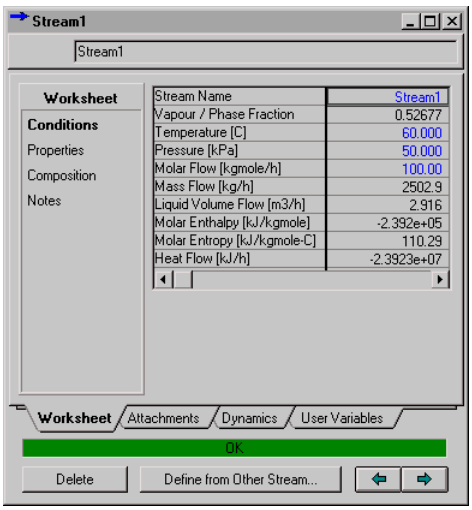
9. Select the **Mole Fractions** radio button prior to the input of values. Enter a Mole Fraction of **1.0** for Water and **1.0** for Methanol.

Figure 1.34



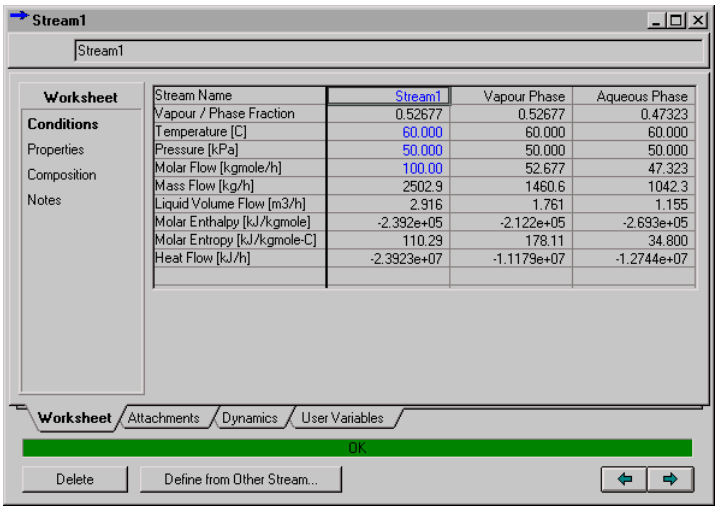
10. Press the ***Normalize*** button. Notice that the **Total** becomes 1.0, as each component is assigned a mole fraction of 0.5.
11. Press the ***OK*** button to return to the stream property view. The mole fractions entered are now present on the **Composition** page.
12. Select the **Conditions** page. The stream will have flashed. Notice the status message shows **OK**.

Figure 1.35



13. Note that it is a two phase stream. Use the horizontal scroll bar to move to the Aqueous and Vapour Phase information of the stream. You can also resize the view horizontally to display all three phases, mixed, aqueous, and vapour. Remember that if the stream property view is Modal, you must press the **Pin** button to make the view Non-Modal before you can resize it. To resize the view, place the mouse pointer on the border and drag horizontally. The resized view is shown next.

Figure 1.36



More information regarding the stream properties can be found on the other pages of the view.

1.3.5 Installing Operations

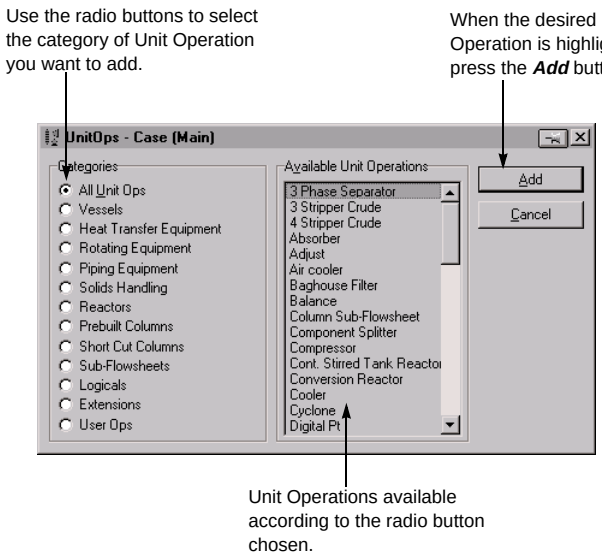
As with Streams, there are various ways to install Unit Operations in HYSYS. Some of these methods immediately open the operation's Property View and some do not.

Direct Property View Access

Method	Description
Menu Bar	New Unit Operations can be installed into the Case via the Menu Bar. Choose the **Flowsheet** option from the Menu Bar, and then select **Add Operation**.
Hot Key	Press **F12**.
Summary View	Open the **Select Summary** view by selecting **Summaries** from the **Tools** option in the Menu Bar. From this view, highlight the Flowsheet in which you will be adding the operation and press the ***View*** button. The **Summary** view will appear, from which you can choose the ***Add*** button in the **Unit Operations** group. You can directly access the **Summary** view by pressing the ***Summary View*** button in the Button Bar
Workbook, Unit Ops Page	From the **Unit Ops** page of the Workbook, press the ***Add UnitOp*** button.

All of these methods will display the **UnitOps** view, listing all the available Unit Operations. Upon selecting the Unit Operation, HYSYS will produce the applicable Property View.

Figure 1.37



Using the Object Palette to install an operation can also give you direct Property View access.

Method	Description
Object Palette	Two methods are available: • Double click on the operation icon in the Object Palette. Click on the desired operation and press the ***Add*** button at the top of the Palette.

Without Direct Property View Access

Object	Description
Workbook	Create a specific tab for the operation type you want to add in the Workbook. For example, add a **Valves** tab to the Workbook for all the valves in the Flowsheet. Once you have the **Operation** tab in the Workbook, you will notice that the default variables for the Operation are shown. In the cell displaying ****New****, enter the name for the new operation.
Object Palette to PFD	Two methods are available: • With the primary mouse button, click on the Icon in the Palette for the operation you want to install. Move the cursor to the PFD area where you want to place the operation, and press the primary mouse button to complete the installation. • With the secondary mouse button, click on the Icon in the Palette for the operation you want to install. Hold the mouse button down and drag the cursor into the PFD. Position the cursor where you want to place the operation and release the button.

Refer to Section 4.5.2 - Editing a Workbook Tab for information concerning the custom Workbook tab.

For a custom Unit Operation Workbook tab, only one type of Unit Operation is allowed.

To access the Property View for an operation, do one of the following:

- On a custom Unit Operation **Workbook** tab (i.e. **Valves** page), double click on one of the Unit Operation's cells.
- Double click on the Icon in the PFD.
- Object inspect (with the secondary mouse button) a cell on a custom Unit Operation Workbook tab. Choose **View** from the menu that appears.
- Object inspect the Icon in the PFD and choose **View Properties**.
- On the Unit Ops tab of the Workbook, highlight the operation and press the ***View UnitOp*** button.
- On the Unit Ops tab of the Workbook, double click on the **Name**, **Object Type** or **Calc**. **Level** cell for the operation.
- Open the Object Status Window (left pane). Double click on the message related to the unit operation.
- Open the Summary view, highlight the operation and press the ***View*** button in the **Unit Operations** group.

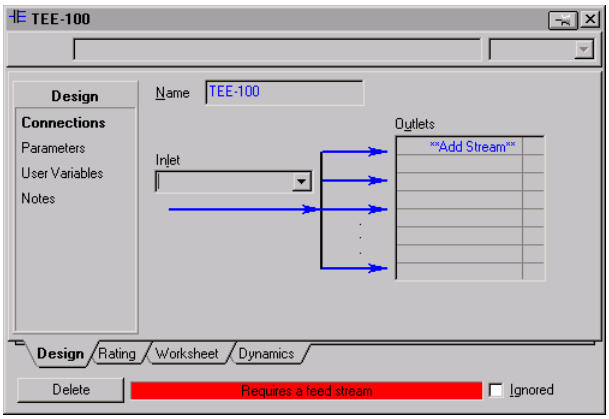
Installing a Unit Operation - Example

Unit Operations can be installed using any of the methods just described. This example demonstrates the installation of a **TEE**. It is assumed that two streams have already been installed, **Stream1** and **Stream2**. The default units are SI.

1. Double click on the **TEE** button in the Object Palette. This opens the Property View, for the **TEE** which has been automatically named **TEE-100**. Focus is currently on the **Inlet** cell.

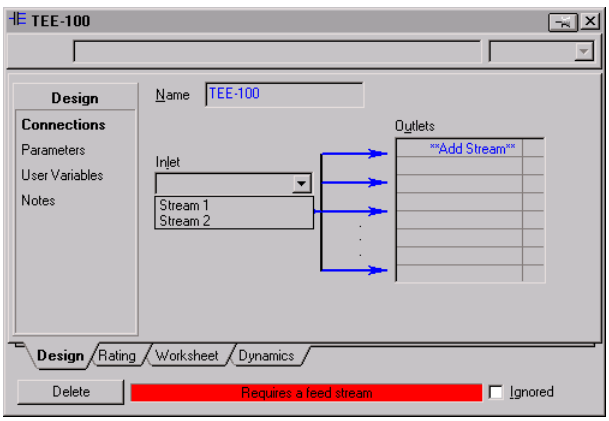
TEE Button

Figure 1.38



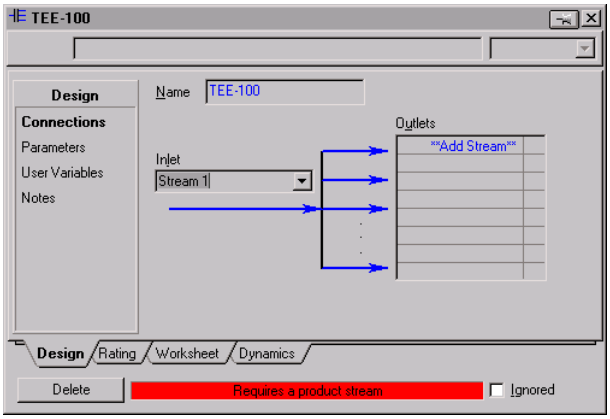
2. You can either type in the Feed Stream name, which can be an existing stream or a new stream, or choose a stream from the drop down list. In this case, use the drop down list to choose one of the streams already installed. With the mouse, click on the down arrow next to the cell to open the drop down list.

Figure 1.39



3. From the drop down list, select **Stream1.** This is now installed as the Feed stream to the **TEE**. The stream name in the **Feed** cell is highlighted and the cursor has not moved from this location. The status message has changed from **Requires a feed stream** to **Requires a product stream**.
4. Move the cursor to the first cell in the **Outlets** group. You can have as many product streams as you want. Again, you can supply streams by typing a name into the cell, or by selecting already installed streams from the drop down list in the Edit Bar. Each method will be used to install one product stream.

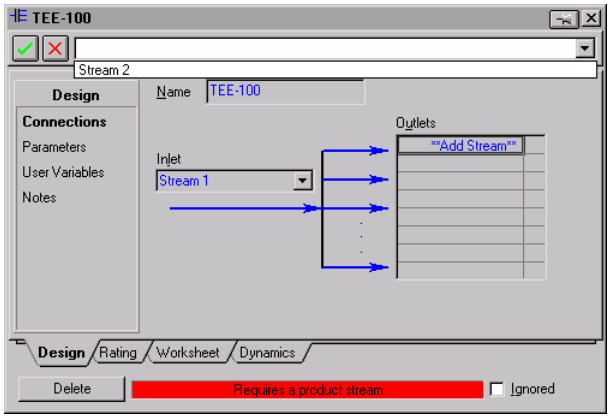
Figure 1.40



You can also select items from the drop down list by simply clicking on them with the primary mouse button.

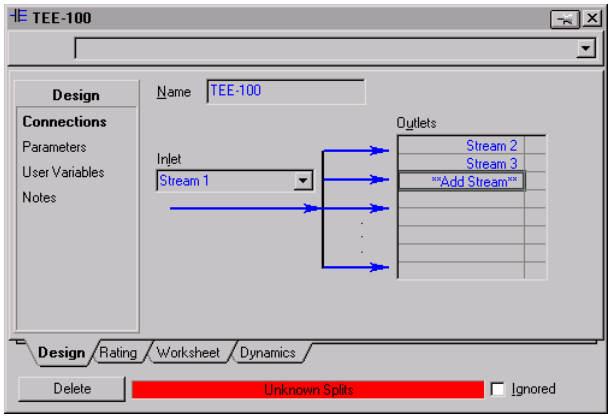
5. Click on the down arrow to the right of the Edit Bar to open the drop down list. The only stream appearing in the drop down list is **Stream2**. Highlight it using the arrow keys and press ENTER to accept it as an input. When you do this, HYSYS places this stream as the first stream in the **Outlets** group.

Figure 1.41



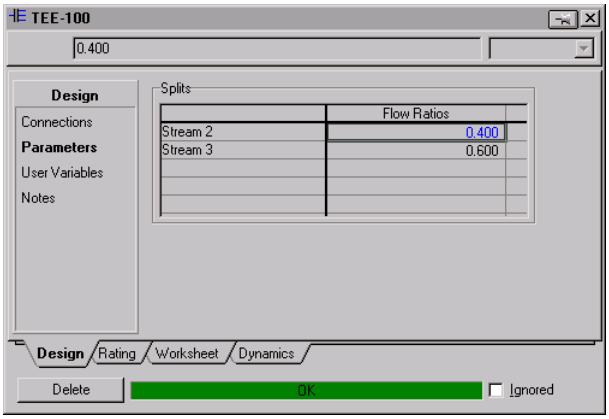
6. In the **Outlets** group, move the highlight to the next cell, which is ready for a new input**.** Now type in the name **Stream3**. Because it is a new stream, HYSYS automatically installs it in the simulation. The **TEE** connections are now complete. The status message displays **Unknown Splits**.

Figure 1.42



7. Go to the **Parameters** page to supply the split fraction for each product stream. The two Product streams, **Stream2** and **Stream3** appear on this page. The sum of the Flow Ratios must equal 1.0, so you only need to supply one ratio. HYSYS will calculate the other ratio by difference. Move the cursor to the **Stream2** Flow Ratio cell and type in **0.40**.

Figure 1.43



8. HYSYS has calculated the Flow Ratio of **Stream3** to be 0.60. In HYSYS, calculated values appear in black and supplied values appear in blue. In this view, 0.40 is shown in blue, and 0.60 in black. Since the **TEE** is being used in Steady State, this is all the information needed. The status message shows **OK**.
9. Press the ***Close*** button to exit the Property View.

1.3.6 Deleting Streams and Operations

Remember, once a stream or operation is deleted, it cannot be recovered.

Property View

You can delete any stream or operation through its Property View by selecting the ***Delete*** button.

Workbook

From the Workbook, you can delete Streams or Operations in various ways, depending on the type of Workbook tab which is being viewed:

- On any **Workbook** tab, double clicking on the operation or stream accesses the Property View for the object. By pressing the ***Delete*** button on the property view, the object will be deleted.
- On any **Workbook** tab but the **Unit Ops** page, select the item with the secondary mouse button which provides the **Object Inspection** menu. Selecting **Delete** from this menu deletes the object.
- Highlight the object **Name** cell and press **DELETE** on the keyboard.
- On the Unit Ops tab only, highlight the operation and press the ***Delete UnitOp*** button.

PFD

To delete streams and operations from a PFD, you can:

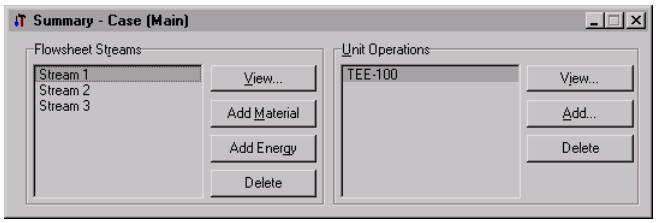
- Double click on the operation or stream accesses the Property View for the object, from which you can delete it by pressing the ***Delete*** button.
- Select the item with the secondary mouse button which provides the Object Inspection menu. Selecting **Delete** from this menu deletes the object.
- Select the object and press **DELETE** on the keyboard.

Summary View

*There are separate **Delete** buttons for the stream and operation groups.*

Once inside the Summary View, you can delete either streams or operations. The **Flowsheet Streams** group lists all the streams present in your case. Similarly, the **Unit Operations** group lists all the operations in your case. Highlight the object you want to delete and then press the ***Delete*** button. You can highlight multiple streams or operations, and then delete them at the same time.

Figure 1.44



No matter which of the above methods you use for deleting objects, HYSYS will prompt you for confirmation before deleting the object. This confirmation requires you to select either Yes or No from the message box that appears.

2 Flowsheet Architecture

2.1 HYSYS Environments 4

2.1.1 Basis Environments 4
2.1.2 Main Simulation Environments 5
2.1.3 Related Environments 7
2.1.4 Advantages of Using Environments 9

2.2 Sub-Flowsheet Environment 10

2.2.1 Sub-Flowsheet Entities 11
2.2.2 Sub-Flowsheet Advantages 11
2.2.3 Multi-Level Flowsheet Architecture 12
2.2.4 Flowsheet Information Transfer 14
2.2.5 Sub-Flowsheet Property View 15
2.2.6 Installing a Sub-Flowsheet 22
2.2.7 Decanter Sub-Flowsheet Example 23

2.3 Templates 32

2.3.1 Template Information 32
2.3.2 Creating a Template Style Flowsheet 36
2.3.3 Installing a Template 37
2.3.4 Example - Refrigeration Loop Template 39

HYSYS has been engineered with a multi-level flowsheet architecture tightly integrated within a framework of *simulation environments*. A direct result of this powerful design is that although HYSYS allows you to interact with an installed sub-flowsheet operation as if it were a simple black box, you can easily delve deeper using the sub-flowsheet's simulation environment when more interaction is required. This intuitive simulation environment framework allows you to focus on the task at hand by providing completely separate Desktops for each environment. It also provides a natural mechanism for HYSYS to be exploited in providing peak computational efficiency for the user. The net result is that potentially complex flowsheets installed as sub-Flowsheet operations behave in a familiar and consistent manner, just like the other "normal" unit operations in HYSYS.

HYSYS also supports a natural extension to the sub-flowsheet concept - the idea of a ***Process Template***. Basically, a Template is a complete Flowsheet that has been stored to disk with some additional information included that pertains to hooking that flowsheet up as a sub-flowsheet operation. Typically a Template is representative of a plant process module or a portion of a process module. The stored Template can subsequently be read from disk and efficiently installed as a complete sub-flowsheet operation any number of times into any number of different simulation cases.

The versatility of the HYSYS sub-flowsheet is extended by the capability of assigning it a separate Fluid Package. For example, this option will enable you to more rigorously model plant utilities such as cooling water and steam circuits as separate flowsheets with dedicated Steam Table property packages.

Column Sub-Flowsheets - A Special Case

There is another feature that the multi-level flowsheet architecture makes possible in HYSYS. This is the extension of HYSIM's traditional or modal style of Column operation into the vastly more flexible and powerful ***Column Sub-Flowsheet Operation*** incorporated in HYSYS. Column sub-flowsheets are a distinct class of sub-flowsheet due to their ability to provide a simultaneous Flowsheet solution. Even though they are different, they are created and accessed much like normal sub-flowsheets, and Column Templates can also be created and later imported into other simulations.

However, the Column sub-flowsheet's property view and the Column's simulation environment are very different, as they are suited specifically for designing Columns rather than general processes.

Although a lot of the general sub-flowsheet information presented in this chapter also applies to the Column sub-flowsheet, the ***Column Sub-Flowsheet Operation*** is discussed specifically and in-depth in Chapter 7 - Column of the **Steady State Modeling** manual.

2.1 HYSYS Environments

The ***environment*** design concept is one of the cornerstones on which HYSYS is built. These environments allow you to access and input information in a certain area ("environment") of the simulation, while other areas of the simulation are put on hold. The other areas won't proceed with steady state calculations until you are finished working in the area of interest. Since the HYSYS integrator is time-step based, the environments have no impact on dynamic calculations.

*The **environments** help you maintain peak efficiency while you are working with your simulation, by avoiding the execution of redundant calculations.*

Separate Desktops are available with each environment. Each of these Desktops includes an appropriate Menu Bar, Button Bar and Home View(s) specifically designed for interaction with their particular environment. The Desktops also remember the views that were open on them, even when their associated environment is not currently active. When moving from one environment to another, they provide a mechanism for quickly and automatically "putting away" what ever views you have open in one environment, and "bringing up" the views that were open in the other environment. This feature is particularly useful when working with large Flowsheets.

The environments in HYSYS can be loosely grouped into two categories for the purposes of discussion: Basis environments and Main simulation environments.

2.1.1 Basis Environments

Within this category, there are the following two types of environments:

- **Simulation Basis Environment**
- **Oil Characterization Environment**

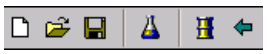
The Simulation Basis Environment

Whenever you begin a HYSYS simulation, you automatically start in the Simulation Basis environment. Here you can create, define and modify Fluid Packages to be used by the simulation's Flowsheets. In general, a Fluid Package contains at minimum a Property Package, as well as

library and/or hypothetical components. Fluid Packages may also contain information such as reactions and interaction parameters.

The Desktop for the Simulation Basis environment is specifically suited to the task at hand as reflected by the available choices of buttons on the Button Bar (Figure 2.1) and the designation of the **Simulation Basis Manager** view as the Home View.

Figure 2.1



The Oil Characterization Environment

The Oil Characterization environment allows you to characterize petroleum fluids by creating and defining Assays and Blends. The Oil Characterization procedure generates petroleum pseudo components for use in your Fluid Package(s). The Oil environment is unique in that it is accessible only from the Simulation Basis environment.

The Desktop for the Oil Characterization environment is very similar in nature to the Desktop available in the Simulation Basis environment. Buttons specific to Generating Oils are provided, and the **Oil Characterization Manager** is the Home View.

Figure 2.2

2.1.2 Main Simulation Environments

These two types of environments are significant with respect to interacting with sub-flowsheets:

- **Main Flowsheet Environment / Sub-Flowsheet Environment**
- **Column Sub-Flowsheet Environment**

You can create sub-flowsheets for all the flowsheets within your simulation.

The Main flowsheet is said to be the *parent* flowsheet for the sub-flowsheets it contains. A sub-flowsheet can also be a parent flowsheet if it contains other sub-flowsheets.

The Main Flowsheet / Sub-Flowsheet Environment

The simulation case's Main Flowsheet environment is the location where you do the majority of your work - installing and defining the streams, unit operations, columns and sub-Flowsheets. This flowsheet serves as the base level or "main" flowsheet for the whole simulation case. Any number of sub-flowsheets may be generated off from this main flowsheet. While there is only one Main Flowsheet environment, each individual sub-flowsheet that is installed has its own corresponding sub-flowsheet environment.

The Desktop for the Main Flowsheet environment contains an extensive Menu Bar and Button Bar designed for building and running simulations. There are two Home Views for the flowsheet - an individual Workbook and PFD.

A sub-flowsheet environment is almost identical to the Main flowsheet environment in that you can install streams, operations and other sub-flowsheets. One difference is that each installed flowsheet in the simulation case will have its own corresponding environment, while there is only one Main flowsheet environment. The other difference is that while you are in a sub-flowsheet environment, Steady State calculations in other areas of the simulation will be put on Hold pending your return to the Main Flowsheet environment.

The Desktop for a sub-flowsheet environment is virtually identical to the Desktop for the Main Flowsheet. There is one minor difference which is the addition to the Button Bar of a ***Parent Simulation Environment*** button.

Figure 2.3

Parent Simulation Environment Button

The Column Sub-Flowsheet Environment

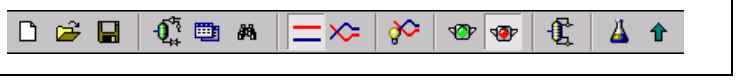
Similar to the sub-flowsheet environment described above, the Column environment is where you install and define the streams and operations contained in a Column sub-flowsheet. Examples of unit operations available for use in a Column sub-flowsheet include tray sections, condensers, reboilers, side strippers, heat exchangers, and pumps. HYSYS contains a number of pre-built Column sub-flowsheet Templates which allow you to quickly install a column of a typical type

There are eleven Prebuilt Columns available in HYSYS.

and then, if necessary, customize it accordingly within its Column environment.

The Menu Bar, Button Bar and Home Views for the Column environment have been designed expressly for designing, modifying, and converging Column sub-Flowsheets. For example, an additional Home View (the Column Runner) has been added and a corresponding menu entry and button on the Button Bar provide access. Even with these changes, a Column environment Desktop still closely resembles the conventional Flowsheet environment Desktop.

Figure 2.4



Column Runner Button

Due to the nature of its solution method, the Column sub-Flowsheet does not support other sub-Flowsheets.

2.1.3 Related Environments

The diagram in Figure 2.5 shows the relationship that exist between the various environments. The arrows indicate the directions in which you would normally move between the environments as you are building a HYSYS simulation. The typical process for building a simulation is as follows:

1. Create a new simulation case, after which you will be in the Simulation Basis environment.
2. Inside the Simulation Basis environment, you can:
   - **Choose a property method and pure components from the HYSYS pure component library.**
   - **Create and define any hypothetical components.**
   - **Define reactions.**

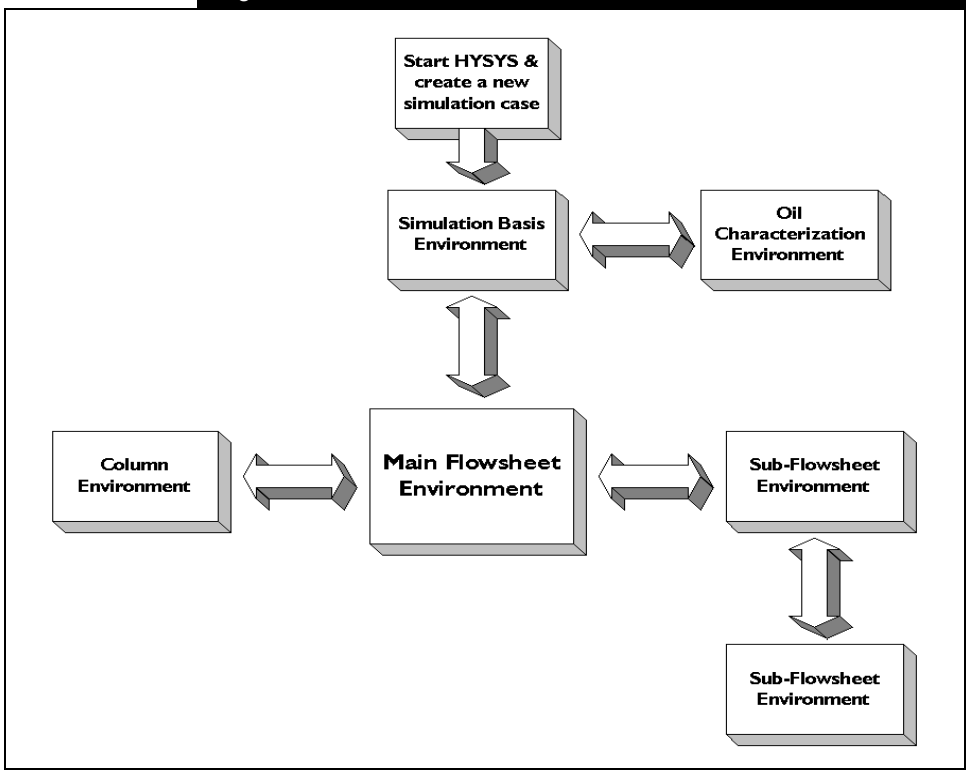
At this point, you have two options. If you have a petroleum fluid to characterize, proceed to step 3. If not, proceed to step 5.

3. Enter the Oil Characterization environment, where you can:
   - **Define one or more Assays and Blends.**
   - **Generate petroleum pseudo components representing the oil.**
4. Return to the Simulation Basis environment.

*In order to access the **Oil** environment you must first be inside the **Simulation Basis** environment.*

5. Enter the Main flowsheet environment, where you can:
   - **Install and define streams and unit operations in the simulation case's Main flowsheet.**
   - **Install Columns operations, Process Templates and sub-flowsheet operations as necessary into the Main flowsheet.**
6. Enter a Column or sub-flowsheet environment when you need to make topological changes, or if you want to take advantage of a sub-flowsheet environment's separate Desktop.

Figure 2.5



Keep in mind that you can move between the flowsheet environments at any time during the simulation. The arrows in the diagram show that the Column and sub-flowsheet environments are accessible only from the Main flowsheet. However, this is only the typical way of moving between the environments. The Navigator allows you to move directly from one flowsheet to any another. The only restriction is that the Oil environment can only be accessed from inside the Simulation Basis environment.

Navigator Button

2.1.4 Advantages of Using Environments

Using environments helps you make the most of your simulation time by eliminating the execution of time-consuming, extraneous calculations.

To illustrate the advantages of the environments approach, consider the creation of a new HYSYS simulation case. When you start HYSYS, you will be placed in the Simulation Basis environment. Here you define a Fluid Package by choosing a property method and components. When finished, you enter the Main flowsheet environment, and proceed to install streams and unit operations.

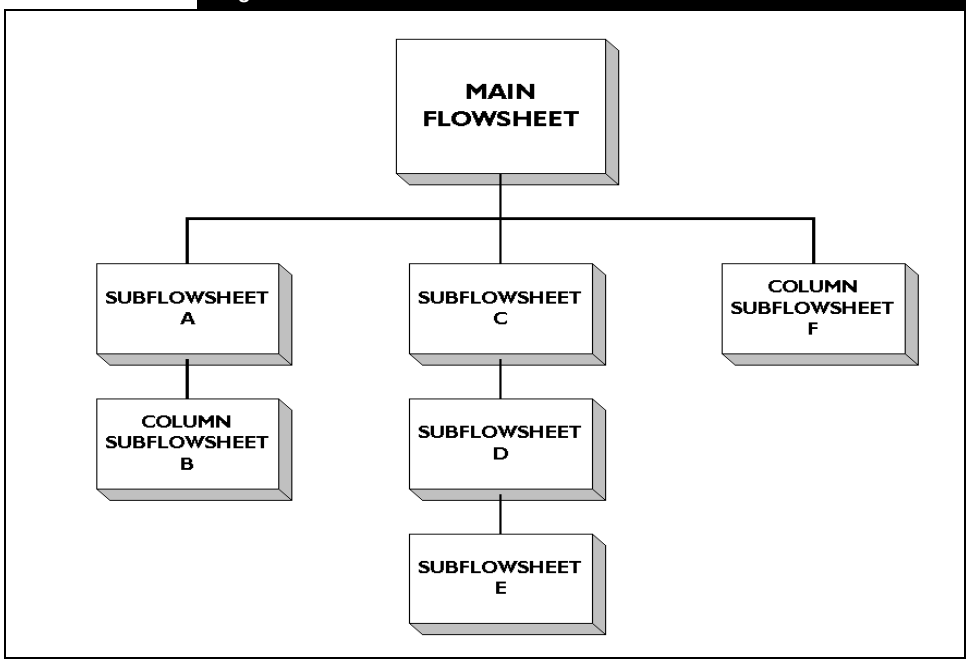
With each time-step, Dynamic calculations proceed from the front to back of the flowsheet in an orderly propagation. This is not affected by the flowsheet environments. Dynamics calculate in a "flat" flowsheet space.

Suppose you now realize that you are missing some components in the Main flowsheet. You can return to the Simulation Basis environment, and all flowsheets will be placed in **Holding** mode until you return. This prevents calculations from taking place until you have made all changes to the Fluid Package. flowsheet calculations will not resume until you instruct HYSYS to do so upon return to the Main flowsheet.

Press the Active (green) button to resume calculations.

For sub-flowsheets, the concept of ***Holding*** Steady State calculations works according to the hierarchy of the flowsheets in the simulation. When you are working inside a particular flowsheet, only that flowsheet and any others below it in the hierarchy will automatically calculate as you make changes. All other flowsheets will hold until you move to their flowsheet's Simulation environment, or one directly above them on the hierarchical tree.

Figure 2.6



Consider the diagram shown in Figure 2.6. Suppose you want to change the number of trays for a column in sub-flowsheet **F**. You would enter the environment for this sub-flowsheet, make the necessary change, then instruct HYSYS to re-calculate the column. As there are no flowsheets below **F** in the hierarchy, all other flowsheets will be on hold while you work on the column. You could continue making changes until you reach a satisfactory solution for **F**. When you return to the Main flowsheet environment, all flowsheets would automatically be re-calculated based on the new sub-flowsheet solution.

Suppose that you now wish to make changes in sub-flowsheet **D**, so you move to its environment. Since **D** is above **E** in the hierarchy, all flowsheets will be on hold except **D** and **E**. Once you reach a new solution for **D**, you might move up to **C**, which will then resume calculations. When you finally return to the Main flowsheet, all other flowsheets (Main, **A**, **B** and **F**) will resume calculations.

If on the other hand you move directly from **D** to **A**, HYSYS will automatically "visit" the Main flowsheet for you so that flowsheet **A** has the most up to date information when you transfer there. Any transfer to a flowsheet not on your "branch" of the tree will force a full recalculation by HYSYS.

2.2 Sub-Flowsheet Environment

The Main simulation environment described in the previous section is one of the cornerstone design concepts upon which HYSYS is built. When combined together with sub-flowsheet capabilities, it defines the basic foundation on which you build a HYSYS simulation. The sub-flowsheet and Column operations embrace the concept of the multi-level flowsheet architecture and provides you with a flexible, intuitive method of building your simulation.

Modelling a large process using several flowsheets helps you better organize your work and manipulate the simulation.

Suppose you are simulating a large processing facility with a number of individual process units. Instead of installing all process streams and unit operations into a single expansive and cumbersome flowsheet, you can simulate each process unit inside its own compact and dedicated sub-flowsheet.

2.2.1 Sub-Flowsheet Entities

Whether the flowsheet is the Main flowsheet of a simulation case, or it is contained in a sub-flowsheet operation, it possesses the following components:

Flowsheet Component	Description
Fluid Package	An independent Fluid Package, consisting of a Property Package, Components, etc. It is not necessary that every flowsheet in the simulation have its own separate Fluid Package. More than one flowsheet can share the same Fluid Package.
Flowsheet Objects	The inter-connected topology of the flowsheet. Unit operations, material and energy streams, utilities etc.
A Dedicated PFD	A HYSYS view presenting a graphical representation of the flowsheet, showing the inter-connections amongst the flowsheet Objects.
A Dedicated Workbook	A HYSYS view of tabular information describing the various types of flowsheet objects in the flowsheet.
A Dedicated Desktop	The PFD and Workbook are home views for this Desktop, but also included are a Menu Bar and a Button Bar specific to either regular or Column sub-flowsheets.

2.2.2 Sub-Flowsheet Advantages

The multi-flowsheet architecture of HYSYS provides a number of technical and functional advantages. The main benefits realized when sub-flowsheets are utilized in a simulation are discussed in the following table:

Capability	Benefit
Multiple Fluid Packages	Each installed sub-flowsheet can have its own Fluid Package within a single simulation case. Note that in some special instances (i.e. a decanter system) a sub-flowsheet can be as small as a single unit operation and its feed and product streams.
Flowsheet Association	Flowsheet association is a design that forces the change of property methods to occur at defined flowsheet boundaries. This ensures that consistent transitions between the thermodynamic basis of the different property methods are maintained and easily controlled.
Simulation Case Organization	Create sub-flowsheets to break large simulations into smaller, easily managed components. This provides an effective means of keeping your simulation concise, while providing the tools (Desktops) to focus your attention on one specific area of the simulation at any time.

There is no limit (except available memory) to the number of flowsheets contained in a HYSYS simulation.

Once a template is installed it will be functionally equivalent to a sub-flowsheet that was created in that simulation case. The only difference is that a sub-flowsheet is unable to be saved to disk and used in another simulation.

Capability	Benefit
Template Creation	Build a process unit as a template style flowsheet (e.g., a refrigeration loop) and save it to disk. You can install this Template into another simulation by simply attaching the necessary feed and product streams as you would any other unit operation. These Templates are fully defined flowsheets, with a property package and components, unit operations, streams and flowsheet specifications.
Nested Flowsheets	Use *nested* flowsheets, i.e. have sub-flowsheets inside other sub-flowsheets. The only restriction on nesting is with columns; that is, you cannot create sub-flowsheet operations inside a Column Operations' sub-flowsheet.

As you become more experienced using HYSYS, you will discover other benefits of the sub-flowsheets. Whether your simulation requires the use of multiple property packages, or involves modelling large and complex processes, using multi-level flowsheeting is the ideal solution.

2.2.3 Multi-Level Flowsheet Architecture

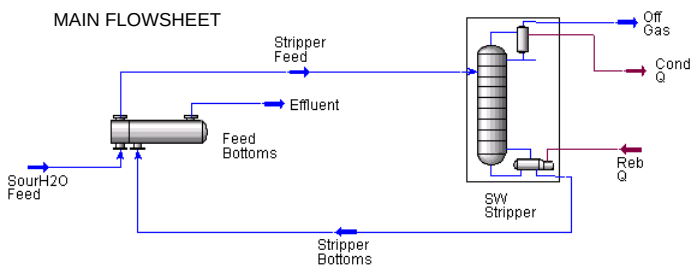
*A Show/Hide option also exists for displaying sub-flowsheet objects on the Main flowsheet PFD. For further details, see **Section 3.8.2 - Accessing Column or Sub-Flowsheet PFDs**.*

From the perspective of the simulation case's Main flowsheet, the sub-flowsheets it contains are discrete unit operations, with feed and product streams. If you are interested only in the feeds to and the products from a sub-flowsheet, you can simply work from the Main flowsheet. However, if you wish to change the topology of the sub-flowsheet, or conveniently view some information about the individual operations in the sub-flowsheet, you can go "inside" the sub-flowsheet to get a more detailed perspective. This is also referred to as "Entering the sub-flowsheet's environment".

This discussion applies equally to sub-flowsheet and Column Operations. For purposes of further discussion, consider the **PFD** of the Main flowsheet for the Sour Water Stripper simulation shown below.

*In the Main flowsheet, the column appears just as any other unit operation (**Figure 2.7**). However the column has its own sub-flowsheet (**Figure 2.8**) that gives you a detailed look at the column's internal streams and operations.*

Figure 2.7



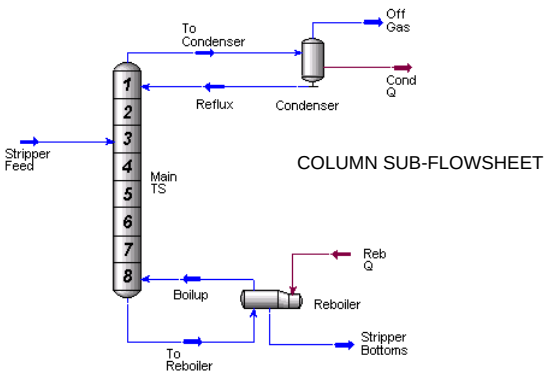
From the simulation environment of the Main flowsheet, the distillation column **SW STRIPPER** appears as any other unit operation, with feed and product streams (e.g., **Feed, Off Gas, Bottoms**). However, the column is also a *sub-flowsheet* with streams and operations of its own. To get a more detailed look at the column, you can go "*inside*" the column sub-flowsheet and examine the streams and operations via the **SW STRIPPER**'s simulation environment. Inside the Column (whose dedicated sub-flowsheet PFD is shown Figure 2.8), the tray section, reboiler and condenser exist as individual unit operations. Similarly, the streams attaching these operations are also distinct (e.g., **To Condenser, Reflux, Boilup, To Reboiler**).

From the perspective of the main flowsheet, the only sub-flowsheet streams of interest are those that attach to it directly. In the case of the Sour Water Stripper, the material streams **Feed**, **Off Gas**, and **Bottoms** and the utility streams **Cooling Water** and **Steam** are the streams of interest. These streams are termed the **Boundary Streams** because they cross out of the main flowsheet's environment into that of the sub-flowsheet, carrying information between parent and sub-flowsheets.

Each sub-flowsheet has its own PFD and Workbook relating only to the information of that flowsheet.

Within the sub-flowsheet environment, a dedicated Workbook and PFD are available for convenient access to the information that pertains only to this sub-Flowsheet. Although information is never hidden or made inaccessible among the various levels of flowsheets in a simulation case, the use of the environments helps organize and focus your simulation efforts in a clear and logical manner.

Figure 2.8



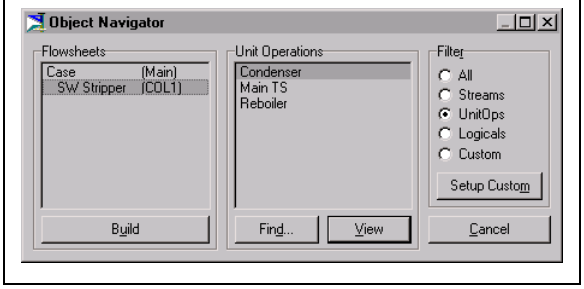
Note that the Simulation environment design basis of HYSYS only allows topological changes to a sub-flowsheet within the Simulation environment for that specific flowsheet.

Multi-Flowsheet Navigation

Object Navigator Button

The multi-flowsheet architecture of HYSYS can be compared to a directory structure. The main flowsheet and its sub-flowsheets are directories and sub-directories, with the streams and operations as the files in that directory. The process information associated with the streams and operations is then, in essence, the contents of the files.

Figure 2.9



HYSYS has special tools, called Navigators, which have been designed to take advantage of this directory-like structure. The **Object Navigator** is shown in Figure 2.9. Within a single window, you can quickly and easily access a stream, operation or process variable in one flowsheet from any other flowsheet in your simulation.

2.2.4 Flowsheet Information Transfer

*By default, the Calculation Level for a sub-flowsheet is set to **2500**, which will ensure that all possible flowsheet calculations in the "Parent" flowsheet are performed before the sub-flowsheet is calculated. This will tend to force the sub-flowsheet to be the last calculation in the chain. In most situations this is the desired behaviour, but may be changed by modifying the sub-flowsheet's calculation Level.*

When you install or create a sub-flowsheet in the Main simulation environment, it appears and behaves as a single operation with one or more feed and product streams. Hence, whenever the values of the streams attached to the sub-flowsheet change, the sub-flowsheet will recalculate just as you expect with any other regular unit operation.

Each of the Parent flowsheet's streams attached to the sub-flowsheet as either a feed or product is associated on a 1:1 basis with a ***boundary stream*** inside the sub-flowsheet. Information flows between the Parent flowsheet and the sub-flowsheet through these associated streams. As a

convenience, when a connection is first established across the boundary, HYSYS automatically renames the sub-flowsheet stream with the name of the stream in the Parent flowsheet. You can override the name reassignment afterwards since the streams on each side of the flowsheet boundary are not required to have the same name. For example, you can have a stream named **To Decanter** in the main flowsheet connected with **Decanter Feed** in a sub-flowsheet.

Components that are not in both flowsheets are ignored, with the remaining compositions re-normalized.

One of the purposes of the sub-flowsheet architecture is to allow the consistent use of different property methods. On each sub-flowsheet's property view, HYSYS allows you to control how stream information is exchanged as it crosses the flowsheet boundary. For example, you can specify that the Vapour Fraction and Temperature (specified or calculated values) of a stream in the Main simulation be passed to the sub-flowsheet. Once this information is inside, the Property Package for the sub-flowsheet then calculates the remaining properties using the transferred composition.

Note that no flash calculations are required for Energy streams. The heat flow will simply be passed between flowsheets.

2.2.5 Sub-Flowsheet Property View

Once a sub-flowsheet operation is installed in a flowsheet, its property view becomes available just like any other flowsheet object. Think of this view as the "outside" view of the "black box" that represents the sub-flowsheet. Some of the information contained on this view is the same as that used to construct a Template type of Main flowsheet. Naturally this is due to the fact that once a Template is installed into another flowsheet, it becomes a sub-flowsheet in that simulation. The sub-flowsheet's property view consists of the following six tabs: **Connections**, **Parameters**, **Transfer Basis**, **Mapping**, **Variables** and **Notes**.

Connections Tab

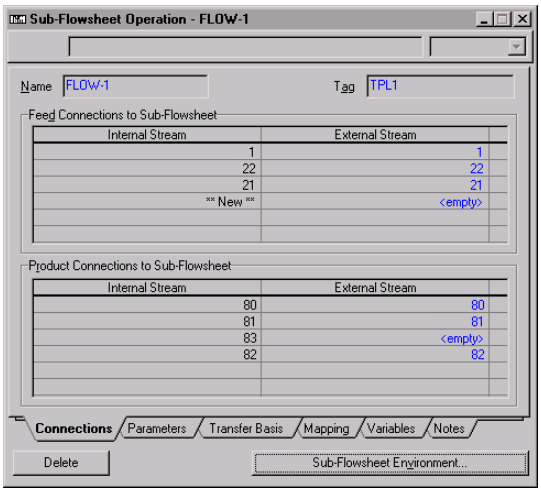
On the **Connections** tab, you can enter the name of the sub-flowsheet, as well as its **Tag** name. All Feed and Product connections are shown.

Flowsheet Tags

These short names are used by HYSYS to identify the flowsheet associated with a stream or operation when that flowsheet object is being viewed outside of its native flowsheet's scope. The default Tag

name for sub-flowsheet operations is **TPL1** (for Template). When more than one sub-flowsheet operation is installed, HYSYS will ensure unique tag names by incrementing the numerical suffix; the sub-flowsheets are numbered sequentially in the order they were installed. For example, if the first sub-flowsheet added to a simulation contained a stream called **Comp Duty**, it would appear as **Comp Duty@TPL1** when viewed from the Main flowsheet of the simulation.

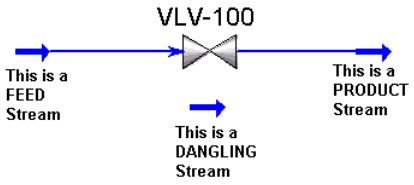
Figure 2.10



Feed and Product Connections

Internal Streams are the Boundary Streams within the sub-flowsheet that can be connected to External Streams in the Parent flowsheet. Internal streams may not be specified on this tab, they are automatically determined by HYSYS. Basically, any streams in the sub-flowsheet that are not completely connected (i.e. are "open ended") can serve as a feed or product, and will appear on this. Note that sub-flowsheet streams that are not connected with any unit ops in the sub-flowsheet will appear in the view as well (and are termed "dangling" streams").

Figure 2.11

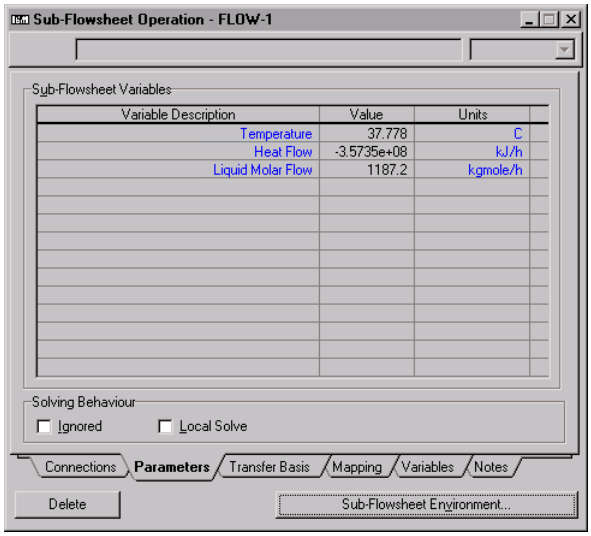


To connect the sub-flowsheet, specify the appropriate name of the external streams, which are in the Parent flowsheet, in the matrix opposite the corresponding internal streams, which are in the sub-flowsheet. The stream conditions are passed across the flowsheet boundary via these connections. Note that it is not necessary to specify an external stream for each Internal Stream.

Parameters Tab

On the **Parameters** tab, you can view the exported sub-flowsheet Variables. This tab is useful when you want to keep track of several key variables without entering the sub-flowsheet environment or adding the variables to the global DataBook. It is also useful in dealing with a sub-flowsheet as a "black box"; the builder of the sub-flowsheet can set up an appropriate **Parameters** tab, and the user of the sub-flowsheet can be quite unaware of the complexities within the sub-flowsheet.

Figure 2.12



These variables are completely live and may display values which have been calculated or specified by the user. If changes to specified values are made here, the sub-flowsheet will be updated accordingly. For each variable, the description, value, and units are shown.

The **Ignore** check box is used to bypass the sub-flowsheet during calculations, just as with all HYSYS unit operations. The **Local Solve** check box is used to solve the **Parameters** tab without solving the entire PFD.

Note that these variable are actually added on the **Variables** tab of the property view, but are viewed in full detail here on the **Parameters** tab.

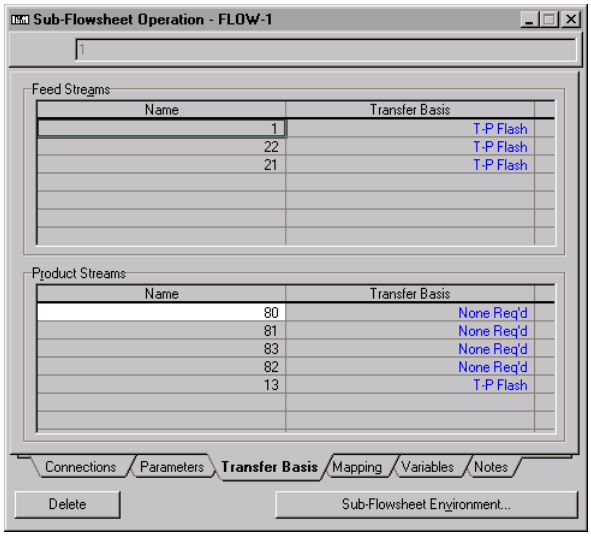
Transfer Basis Tab

The Transfer Basis is also useful in controlling VF, T or P calculations in Column sub-flowsheet boundary streams with close boiling or nearly pure compositions.

The transfer basis for each Feed and Product Stream is listed on the **Transfer Basis** tab. The transfer basis only becomes significant when the sub-flowsheet and Parent flowsheet's Fluid Packages consist of different property methods. The transfer basis is used to provide a consistent means of switching between the differing basis of the various property methods:

Transfer Basis	Description
T-P Flash	The Pressure and Temperature of the Material stream are passed between flowsheets. A new Vapour Fraction will be calculated.
VF-T Flash	The Vapour Fraction and Temperature of the Material stream are passed between flowsheets. A new Pressure will be calculated.
VF-P Flash	The Vapour Fraction and Pressure of the Material stream are passed between flowsheets. A new Temperature will be calculated.
P-H Flash	The Pressure and Enthalpy of the Material stream are passed between flowsheets.
User Specs	You define the properties passed between flowsheets for a Material stream.
None Required	No calculation is required for an Energy stream. The heat flow will simply be passed between flowsheets.

Figure 2.13

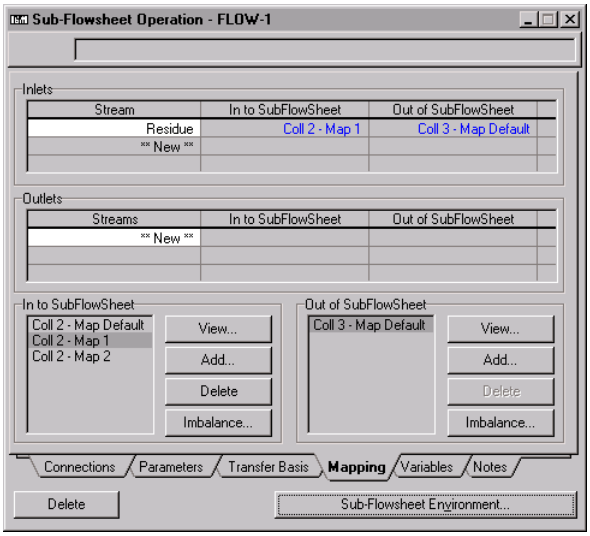


Mapping Tab

On the **Mapping** tab you can map fluid component composition across Fluid Package boundaries. Composition values for individual components from one Fluid Package can be mapped to a different component in an alternate Fluid Package. This is especially useful when dealing with hypothetical oil components where like components from one fluid package can be mapped across the subflowsheet boundary to another fluid package. Using a component map, mass balance can be conserved.

At lease two previously defined Fluid Packages are required to perform a component mapping which is defined as a *collection*. The collection is created in **Components Maps** tab found in the **Simulation Basis Manager** (refer to **Section 5.2 - Component Maps Tab** in the **Simulation Basis** manual for more information).

Figure 2.14



Component Maps

The inlet and outlet component maps are listed in the **In to SubFlowSheet** and **Out of SubFlowSheet** group boxes respectively. The buttons located next the component map lists perform the following functions.

Button	Function
View	Allows you to view and edit the component map by accessing the **Component Map Property** view. Refer to **Section 5.3 - Component Map Property View** in the **Simulation Basis** manual for more information).
Add	Allows you to define a new component map using the **Component Map Property** view. Refer to **Section 5.3 - Component Map Property View** in the **Simulation Basis** manual for more information).
Delete	Allows you to delete a component map.
Imbalance	Opens the **Untransferred Component Info** view, allowing you to confirm that all of the components have been transferred in to the subflowsheet.

Inlet and Outlet Streams

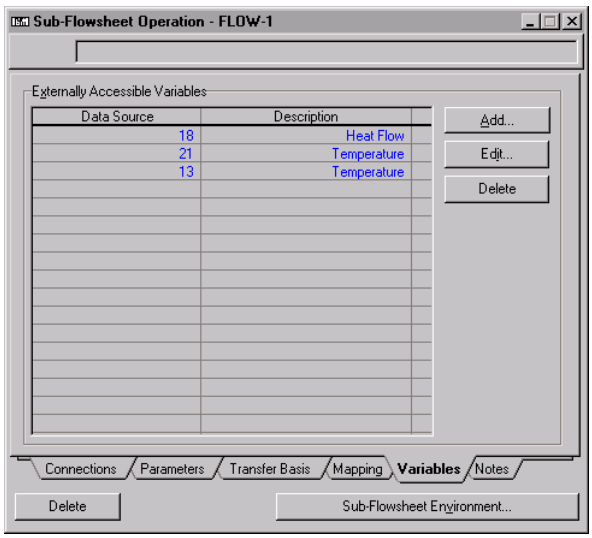
To attach a component map to inlet and outlet streams, simply specify the name of the inlet component map in the **In to SubflowSheet** cell and the name of the outlet component map in the **Out of SubFlowSheet** cell of the desired stream.

Variables Tab

The **Variables** tab of the Main flowsheet's property view is for creating and maintaining the list of **Externally Accessible Variables**. Although you can access any information inside the sub-flowsheet using the Variable Navigator, this feature allows you to target key process variables inside the sub-flowsheet and have their values displayed on the property view. Then you can conveniently view this whole group of information directly on the sub-flowsheet's property view in the Parent flowsheet.

To add variables to this tab, select the ***Add*** button. The Variable Navigator will be available, where you can select the flowsheet object and variable. On the navigator you may over-ride the default variable description and provide a better description if you wish. Note that these variables appear on the **Parameters** tab of this view in a format suitable for viewing. Refer to Chapter 1 - Interface for details on the Variable Navigator.

Figure 2.15



2.2.6 Installing a Sub-Flowsheet

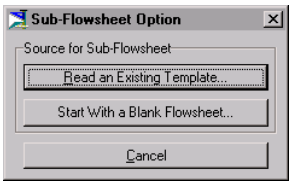
Sub-Flowsheet Button

To install a sub-flowsheet, choose **Add Operation** from the flowsheet Menu or press **F12**, and select **Sub-flowsheet**. Alternatively, you could select the ***Sub-Flowsheet*** button on the Object Palette.

Once you initiate the installation of a sub-flowsheet, you must specify how you would like to proceed. Your options are:

- reading an existing template
- starting with a blank flowsheet.

Figure 2.16



Reading an Existing Template

If you want to use a previously constructed Template that has been saved on disk, select the ***Read an Existing Template*** button. The process of installing a Process Template is covered in detail in Section 2.3.2 - Creating a Template Style Flowsheet.

Starting with a Blank Flowsheet

Starting with a blank flowsheet is a good option if you are just creating a small sub-flowsheet, otherwise you should consider creating a full template flowsheet so it may be re-used in future simulations.

If you select ***Start with a Blank Flowsheet,*** HYSYS will install a sub-flowsheet operation containing no unit operations or streams. As with other unit operations, the property view of the sub-flowsheet will be opened, and you will be placed on the **Connections** tab. At this point, there will be no feed or product connections (***Boundary Streams***) to the sub-flowsheet. However, you can connect feed streams in the **External Stream** column by typing directly in the cell or by making a selection from the Edit Bar drop down list. This will either create a new stream in the main flowsheet or use an existing stream. When an external feed connection is made in the aforementioned way, its equivalent stream is created inside the sub-flowsheet environment.

If you choose ***Start with a Blank Flowsheet***, the sub-flowsheet you create will not be available to use in any other simulation case you may

happen to create in the future. If you think you would like to be able to re-use the sub-flowsheet at a later date, perhaps you should consider creating a Template instead. This is covered in detail in Section 2.3.2 - Creating a Template Style Flowsheet.

In order to fully define the flowsheet, you have to enter the sub-flowsheet's environment. To do so, select the ***Sub-Flowsheet Environment*** button on the property view to transition to the sub-flowsheet's environment and its dedicated Desktop. Construction of the flowsheet can occur in the usual manner, and when you return to the Parent environment, you will be able to connect the sub-flowsheet's Boundary Streams to streams in the Parent flowsheet, and everything will solve. An example of this process follows.

2.2.7 Decanter Sub-Flowsheet Example

When simulating an azeotropic distillation column with a decanter, two property methods are often preferred. A property method optimized for Liquid-Liquid equilibrium is often desired for the decanter, while a property method optimized for Vapour-Liquid equilibrium is desired for the main tower. Utilizing a sub-flowsheet for the decanter is the ideal way to solve these types of simulations.

The following example of the dehydration of an azeotropic ethanol/water mixture using benzene as an entrainer illustrates the use of a sub-flowsheet as a means of providing a different property method for the decanter.

Setting up the Fluid Packages

Two Fluid Packages are required for this example:

Name [Property Package]	Components
VLE-Basis [UNIQUAC]	Ethanol, H2O, Benzene
LLE-Basis [NRTL]	Ethanol, H2O, Benzene

1. The first Fluid Package is named **VLE-Basis.** Select the **UNIQUAC** activity model and the components **Ethanol, H2O** and **Benzene**.
2. On the **Fluid Pkgs** tab of the **Simulation Basis Manager** view, press the ***Copy*** button to copy **VLE-Basis** to create the new Fluid Package, **LLE-Basis**.

*To save time you can copy the first Fluid Package using the **Copy** button on the **Fluid Pkgs** tab of the Simulation Basis Manager and then make the changes.*

3. Open the view for the **LLE-Basis**. Change the property method to **NRTL** and then move to the **Binary Coeffs** tab.
4. Select the ***UNIFAC LLE*** radio button and press the ***All Binaries*** button. This will re-regress all interaction parameters such that they are set up for two liquid phase predictions.

Define the Feed Streams

Two feed streams are required for the dehydrator; the estimated reflux from the decanter and the almost azeotropic raw feed mixture of ethanol and water.

1. Create two streams: **Feed** and **Reflux**. Specify the properties as defined in the following table.

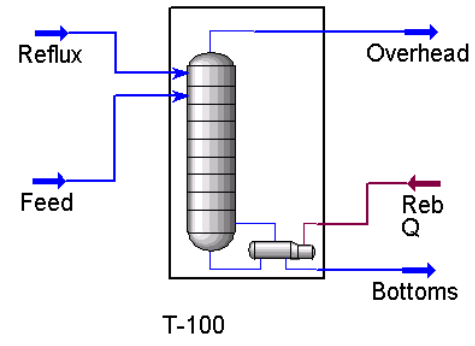
Stream Name	Feed	Reflux
Temperature [C]	78	50
Pressure [atm]	1	1
Mass Flow [kg/hr]	3000	15000
Comp Mole Frac [Ethanol]	0.88	0.50
Comp Mole Frac [Water]	0.12	0.00
Comp Mole Frac [Benzene]	0.00	0.50

Install the Dehydrator Column

Install a 30-stage **REBOILED ABSORBER**, which will be used to model a large portion of the dehydrator distillation column. Further on, in the example, a separate condenser and decanter will be modelled in a sub-flowsheet.

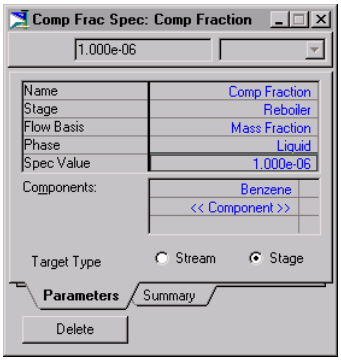
Reboiled Absorber Button

Figure 2.17



1. Install the connections as shown in the PFD of Figure 2.17. The **Reflux** enters on **tray 1** and the **Feed** stream on **tray 5**.
2. The pressure profile is at **1 atm**.
3. In the **Damping** group on the **Solver** page of the **Parameters** tab, enter a **Fixed Damping Factor** of **0.5**.
4. Install an impurity specification of **1.0e-06 benzene mass fraction** in the **Bottoms** stream. See Figure 2.18.

Figure 2.18



5. Press the ***Run*** button on the Column property view. The tower will converge and an ethanol purity in excess of 99.5% will be attained in the **Bottoms** stream.

Install a Sub-Flowsheet Operation

Perform the following steps to install the decanter sub-flowsheet:

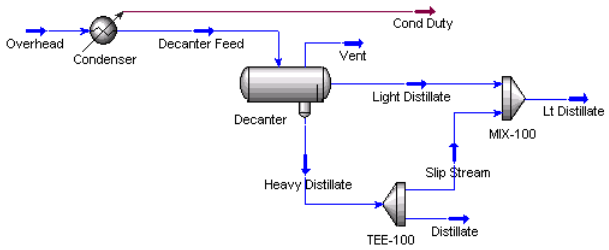
Sub-Flowsheet Button

1. Begin the sub-flowsheet operation installation by double clicking on the Object Palette's sub-flowsheet icon, or by pressing **F12** and selecting **Sub-Flowsheet** from the **UnitOps** view.
2. The **Sub-Flowsheet Option** view will appear. Select ***Start With a Blank Flowsheet***.
3. The **Sub-Flowsheet** property view will open to the **Connections** tab. Change the default **Name** to **Decanter System**.

Build the Decanter Flowsheet

1. In order to create the flowsheet, you have to enter the sub-flowsheet's environment. Selecting the ***Sub-Flowsheet Environment*** button on the property view for transition to the sub-flowsheet's environment and its dedicated Desktop.

Figure 2.19



2. The environment indicator on the Button Bar will indicate that you are now in the **Decanter System** (**TPL1**). The Desktop for the decanter flowsheet appears, and either the Workbook or the PFD will be on the Desktop (depending on the preferences settings).

Install a **COOLER** operation by double clicking on the Cooler icon in the Object Palette, or by pressing **F12** and selecting **Cooler** from the **UnitOps** view. Supply the following information:

COOLER (Condenser)		
Tab [Page]	**Input Area**	**Entry**
Design [Connections]	Name	Condenser
	Inlet	Condenser Feed
	Outlet	Decanter Feed
	Energy	Cond Duty

COOLER (Condenser)		
Tab [Page]	**Input Area**	**Entry**
Design [Parameters]	Delta P	5 psi

3. Specify the Vapour Fraction of **Decanter Feed** to be 0.0000.
4. Install a **3-PHASE SEPARATOR** operation. Supply the following information:

Tab[Page]	Input Area	Entry
Design [Connections]	Name	Decanter
	Feeds	Decanter Feed
	Vapour	Vent
	Light Liquid	Light Distillate
	Heavy Liquid	Heavy Distillate

5. As shown in the PFD of Figure 2.19, some of the **Heavy Distillate** is usually mixed in with the **Light Distillate** to create the **Reflux**. This is typically done to promote better column operability. Install a **TEE** and **MIXER** to provide this functionality:

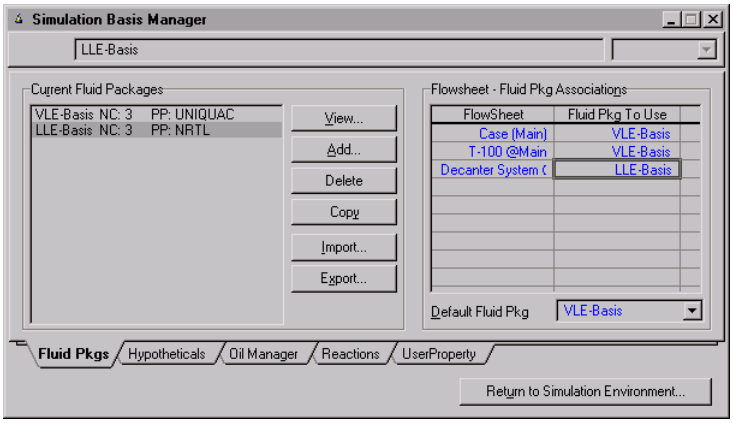
Operation	Tee	Mixer
Inlet Stream(s)	Heavy Distillate	Light Distillate Slip Stream
Outlet Stream(s)	Distillate Slip Stream	Reflux
Splits	Distillate - 0.9 Slip Stream - 0.1	n/a

Assign the LLE-Basis Fluid Package

1. Select the ***Simulation Basis Environment*** button from the Main Button Bar.

Simulation Basis Environment Button

Figure 2.20



2. In the **Flowsheet - Fluid Pkg Associations** group box, change the Fluid Package assigned to the Decanter System (TPL1) flowsheet from the default of "VLE-Basis" to the one designed just for the decanter: "LLE-Basis". This is shown in Figure 2.20.
3. Return to the Decanter's sub-flowsheet environment by selecting the ***Return to Simulation Environment*** button on the **Simulation Basis Manager** view. Select **No** when queried as to whether HYSYS calculations should be put on hold.

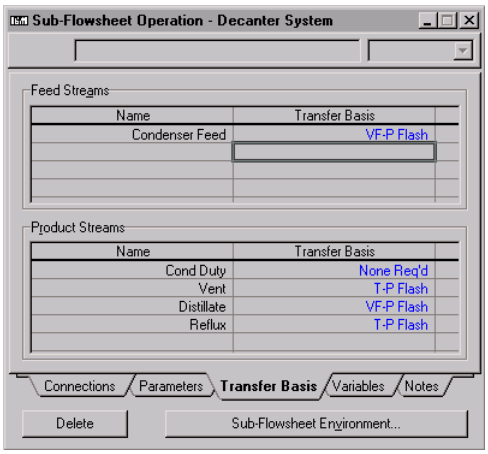
Specify Proper Transfer Basis

Parent Simulation Environment Button

1. Select the ***Parent Simulation Environment*** button on the Main Button Bar to return to the Main flowsheet. Once there, open the property view for the Decanter sub-flowsheet.
2. On the **Connections** tab, you should see Condenser Feed as the Boundary Stream available in the feeds matrix. The following Boundary Streams should be available as products: **Cond Duty**, **Vent**, **Distillate**, and **Reflux**.
3. The decanter's condenser should be dealing with a dew point feed and the decanter itself is producing bubble point products. Therefore, it is evident that the default Transfer Basis of **T-P** will not be suitable as these are not the proper stream variables for preserving the dew point/bubble point condition of the feed/product streams when the thermodynamic basis changes across the flowsheet boundary. Select the **Transfer Basis** tab to begin the modification procedure.

4. Choose the following Transfer Basis for the feeds and products: Condenser Feed VF-P, Reflux T-P (not necessarily at BP because of the mixed in Heavy Distillate slipstream), Distillate VF-P.

Figure 2.21



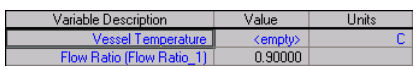
Export Key Flowsheet Variables

1. It is very convenient to view or specify key process variables of the decanter from the Main flowsheet environment. Go to the **Variables** tab of the property view.
2. Press the ***Add*** button on this tab to summon the **Variable Navigator** where you will select the appropriate variables from the sub-flowsheet and supply the appropriate variable descriptions. The variables are defined in the following table:

Field	Variable 1	Variable 2
Flowsheet	Decanter System	Decanter System
Object	Decanter	TEE-100
Variable	Vessel Temperature	Flow Ratio
Variable Specifics	n/a	Flow Ratio_1

3. On the **Parameters** tab, the two variables which you have exported will appear.

Figure 2.22



Once the main flowsheet has converged, the **Parameters** tab will appear as shown in Figure 2.23.

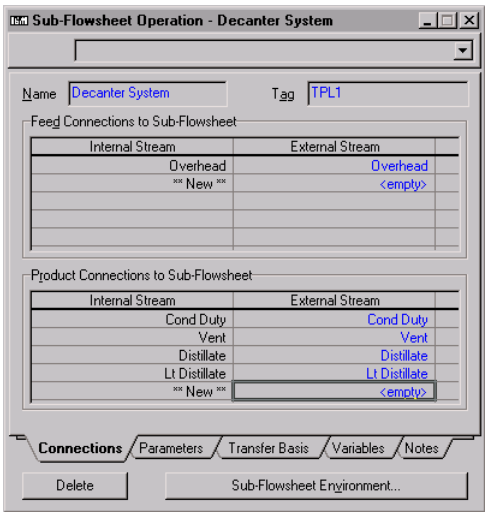
Figure 2.23

Variable Description	Value	Units
Vessel Temperature	52.835	C
Flow Ratio (Flow Ratio_1)	0.90000	

Attach the Main Flowsheet Streams to Decanter

1. Switch back to the **Connections** tab on the property view.
2. Connect the absorber overhead stream as the feed connection. Note that the **Internal Stream** name will change to **Overhead** once the External Stream name is selected. The sub-flowsheet will have enough information to solve once the Overhead feed is connected.
3. Supply an appropriate stream name for the condenser duty.
4. Supply names for the Vent, Reflux and Distillate product streams. Input **Lt Distillate** in the External Stream cell next to Reflux. Since the decanter has solved, the information will be transferred to the external streams as soon as they are connected. See Figure 2.24.

Figure 2.24



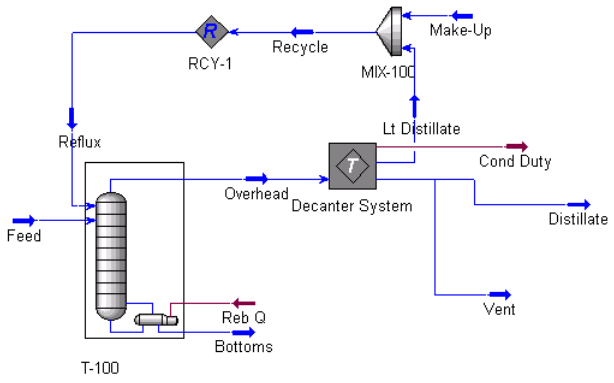
5. Note that you must install a **RECYCLE** operation, a **MIXER**, and usually some sort of a make-up stream is required to offset any solvent losses in the products:

Stream Name	Make-Up
Temperature [C]	20
Pressure [atm]	1
Mass Flow [kg/hr]	1
Comp Mole Frac [Ethanol]	0.00
Comp Mole Frac [Water]	0.00
Comp Mole Frac [Benzene]	1.00

Field	MIxer	Recycle
Inlet Stream(s)	Lt Distillate Make-Up	Recycle
Outlet Stream	Recycle	Reflux

Once the recycle has converged, the sub-flowsheet's property view can be opened to the **Parameters** tab and the Heavy Distillate's slipstream fraction varied as desired. After each change, the whole column/ decanter recycle process will automatically converge on the new answer corresponding the specified slipstream fraction. See **Figure 2.25** for the completed PFD.

Figure 2.25



2.3 Templates

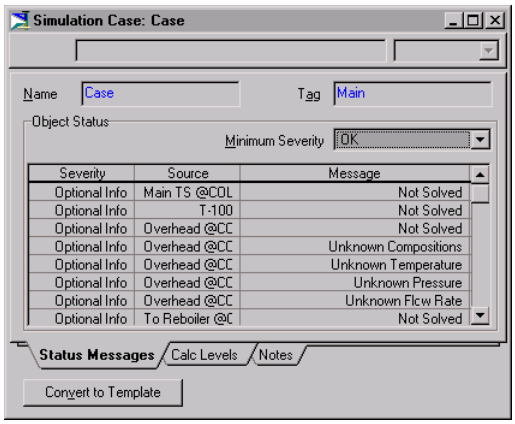
A Template flowsheet is simply a normal HYSYS flowsheet with some additional information contained in its Main properties and a different file extension used when it is stored to disk (*.TPL versus the normal *.HSC). The different file extension is employed mainly for organizational purposes.

2.3.1 Template Information

The Template information for the flowsheet is accessed through the Main flowsheet's property view (**Simulation - Main Properties** or CTRL M). The first two tabs of this property view are the same as for any simulation case. However, there are three additional tabs that are exclusive to Templates.

The tabs become available once the standard simulation case is converted to a template. This is accomplished by selecting the ***Convert to Template*** button on the bottom of the Main properties view. Once the button has been selected, and the extra tabs appear, the button will no longer be visible.

Figure 2.26

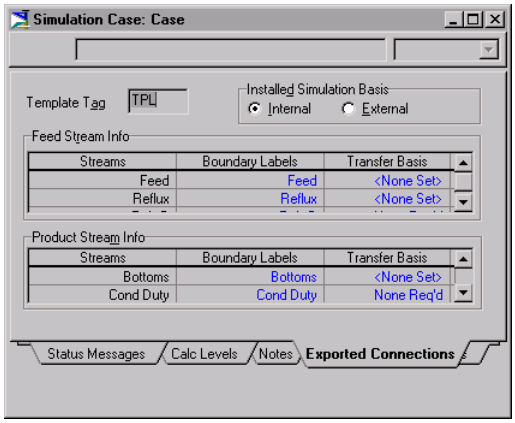


These extra tabs contain all of the same information available on the property view of an installed sub-flowsheet operation as well as some additional information. These extra parameters allow the flowsheet to be treated as a "black box" and installed as a sub-flowsheet operation with the same ease and in the same manner as a normal unit operation.

Exported Connections Tab

On the **Exported Connections** tab, you can enter the Template Tag and select the Installed Simulation Basis. All Feed and Product connections are also shown on this tab.

Figure 2.27



Template Tag

Flowsheet **Tags** are short names used by HYSYS to identify the flowsheet associated with a stream or operation when that flowsheet object is being viewed outside of its native flowsheet's scope. The default Tag name for sub-flowsheet operations is **TPL1** (for Template). When more than one sub-flowsheet operation is installed, HYSYS will ensure unique tag names by incrementing the numerical suffix in a manner similar to HYSYS auto-naming unit operations; they are numbered sequentially in the order they were installed. For example, if the first sub-flowsheet added to a simulation contained a stream called **Comp Duty**, it would appear as **Comp Duty@TPL1** when viewed from the Main flowsheet of the simulation.

Installed Simulation Basis

Once a Template is installed the resulting Fluid Package association may be over-ridden in the Simulation Basis Manager at any time.

When a Template is read into a simulation case, its associated Fluid Package is added to the list of Fluid Packages in the Simulation Basis Manager. The Installed Simulation Basis gives the Template builder the choice of using its own internal Fluid Package, or the same Fluid Package of the Parent flowsheet in which it is installed. This only affects what happens at the time the template is first installed.

Feed and Product Stream Info

*A stream that appears on the **Exported Connections** tab does not necessarily have to be connected.*

All streams in the flowsheet Template that are not completely connected, i.e., are only a feed to a unit operation, or a product from a unit operation are designated as ***Boundary Streams***, and will appear in the appropriate Group Box. Boundary Streams may not be chosen to appear on this tab, they are automatically determined by HYSYS. These are the streams that you will be connecting to when the Template is installed in a flowsheet.

For each stream appearing in either the Feed Stream or Product Stream matrices, you can specify the **Boundary Label** and **Transfer Basis**.

A **Boundary Label** is what is used to describe the name of the feed and product connections. This is not the name of the streams, but rather the function of the streams, i.e. if you are using a numerical standard for stream numbering, the feed stream inside the template could be "1", but its feed label could be "HP Feed". This allows you to provide descriptive feed and product stream labels, much like the built-in unit operation property views used on their connection tabs. By default it will just assume the name of its corresponding boundary stream in the Template.

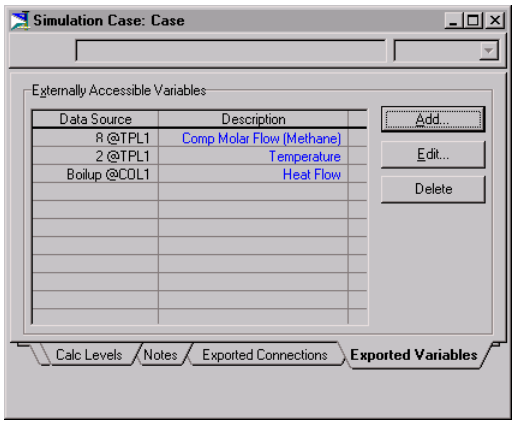
The **Transfer Basis** is used for feed and product streams as they cross the flowsheet boundary. The Transfer Basis becomes significant only when the sub-flowsheet and parent flowsheet Property Packages are different. When there are differing Fluid Packages in the two flowsheets (Parent and sub-flowsheet) you may wish to specify what stream properties will be used to calculate the stream on the other side of the boundary. The Transfer Basis is used to provide a consistent means of switching between the differing basis of the various property methods:

Flash Type	Description
T-P Flash	The Pressure and Temperature of the Material stream are passed between flowsheets. A new Vapour Fraction will be calculated.
VF-T Flash	The Vapour Fraction and Temperature of the Material stream are passed between flowsheets. A new Pressure will be calculated.
VF-P Flash	The Vapour Fraction and Pressure of the Material stream are passed between flowsheets. A new Temperature will be calculated.
None Required	No calculation is required for an Energy stream. The heat flow will simply be passed between flowsheets.

Exported Variables Tab

The last tab of the Main flowsheet's property view is for creating and maintaining the list of Exported Variables. Although you can access any information inside the sub-flowsheet using the Variable Navigator, this feature allows you to target key process variables inside the sub-flowsheet and have their values displayed on the property view. Then, when the Template is installed, you can conveniently view this information directly on the sub-flowsheet's property view in the Parent flowsheet. This is very useful for "black box" treatment, as all the important specifications for the operation of the sub-flowsheet may be brought together and documented in this one location. You may never have to enter the sub-flowsheet environment to get the template "working" or adjusted to your needs.

Figure 2.28



To add variables to this tab, select the ***Add*** button. This will bring up the Variable Navigator for you to select the flowsheet object and variable. On the navigator you may over-ride the default variable description and provide a better description. When you install this template into another case, these variables will appear in the **Parameters** tab of the sub-flowsheet property view (see **Section 2.2.5 - Sub-Flowsheet Property View** for information on the **Parameters** tab).

Note that there is no difference between a template flowsheet and a normal flowsheet, except the additional information mentioned above, and the use of different file extensions. A template flowsheet can be read in as the Main flowsheet in a simulation case if necessary - you will just get a warning message and the extra information will be ignored.

2.3.2 Creating a Template Style Flowsheet

Note that you cannot create a Template from just parts of a Main Flowsheet. You will have to delete any unwanted streams and operations from the Main Flowsheet before saving it out to disk. Of course it can be saved with a different name, preserving your original simulation case.

Any Main flowsheet may be used as the base for a Template. It is simply a matter of toggling it to being a template style flowsheet, supplying the extra information necessary for it to be installable in any simulation case, and saving it out to disk.

Note that there are two ways to toggle a flowsheet to being of a Template Style:

- If you *have already created* a New Case (not a New Template) and now decide you want to save it is a Template after it has been built, or if you have an existing case on disk that you would like to use as a template:

1. Access the cases Main flowsheet's Property view (**Simulation - Main Properties** or CTRL M) and select the ***Convert to Template*** button.
2. Set the Template Tag, Installed Simulation Basis and other optional template information if required. See **Section 2.3.1 - Template Information** for a discussion on Template Information.
3. When you save the simulation, it will be saved as a template.

You cannot create a Template from an existing sub-Flowsheet that is part of a larger simulation.

- **If you *know ahead of time* that the flowsheet is to be a Template, you can begin by creating a new Template:**

1. From the **File** menu, select **New**, then **Template**.
2. Follow the standard procedure for building your simulation.
3. Access the Main flowsheet's property view (**Simulation - Main Properties** or CTRL M) and set the Template Tag, Installed Simulation Basis and other optional template information if required. See **Section 2.3.1 - Template Information** for a discussion on Template Information.
4. When you save the simulation, it will be saved as a template.

Keep in mind that you can have multiple simulation cases in memory, so you can create a new Template as part of your current session and then install it into your original Simulation case.

HYSYS will automatically save the Template in the Templates directory as a template file (***.TPL**). The default path for the Templates directory is set according to the HYSYS preferences. As shipped, the default directory is **HYSYS\TEMPLATE**.

Note that a combination of flowsheets may be in your template, i.e. a Main flowsheet and one or more sub-flowsheets. Likewise, more than one Fluid Package maybe included in the template if they are associated with a flowsheet at the time the template is saved to disk.

2.3.3 Installing a Template

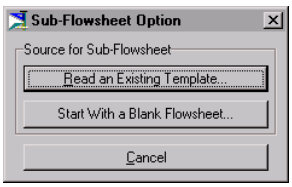
To install a Template you follow the same basic procedure as when installing a sub-flowsheet. Choose **Flowsheet-Add Operation** from the menu or press **F12**, and select **Sub-Flowsheet**. Alternatively, you could select the ***Sub-Flowsheet*** button on the flowsheet Object Palette.

Sub-Flowsheet Button

Once you initiate the installation a sub-flowsheet, you will be queried with a **Sub-Flowsheet Option** view as to how you wish to proceed. Your options are:

- **Reading an existing Template**
- **Starting with a blank Flowsheet**

Figure 2.29



*If you choose **Start with a Blank Flowsheet**, the sub-flowsheet you create will not be available to use in any other simulation case you may happen to create in the future.*

If you think you would like to be able to re-use the sub-flowsheet at a later date, you should consider creating a Template instead.

- **Starting with a Blank Flowsheet**

If you do not want to bother constructing a Template, select the ***Start with a Blank Flowsheet*** button. The process of creating a sub-flowsheet with a blank initial flowsheet operation was covered in detail previously in Section 2.2.6 - Installing a Sub-Flowsheet.

This is a good option if you are just creating a small sub-flowsheet, otherwise you should consider creating a full template flowsheet so it may be re-used in future simulations.

Reading an Existing Template

To install a Template style flowsheet, select the ***Read an Existing Template*** button. HYSYS will look in the Templates directory for available template files (***.TPL**). The default path for the Templates directory is set according to the HYSYS preferences. As shipped, the default Templates directory is **HYSYS\TEMPLATE**.

HYSYS includes a sample Process Template for trial purposes. It is called REFRIG.TPL and is identical to the example created in the next section: **Section 2.3.4 - Example - Refrigeration Loop Template.**

If there are sub-flowsheets in the template, they will be installed as sub-flowsheets underneath the new sub-flowsheet operation. In other words, everything in the template is "shifted down" at least one level.

When a file is selected, HYSYS installs any Fluid Packages in the Template into the Simulation Basis Manager. The Main flowsheet contained in the Template is then installed as a new sub-flowsheet unit operation in the current flowsheet you are in. After the flowsheet(s) have been transferred into the simulation case, a Fluid Package will be selected for the sub-flowsheet based on the ***Installed Fluid Package*** setting used in the Template. Once HYSYS finishes all this book keeping, you will be placed on the **Connections** tab of the sub-flowsheet property view where you can begin hooking up the Template.

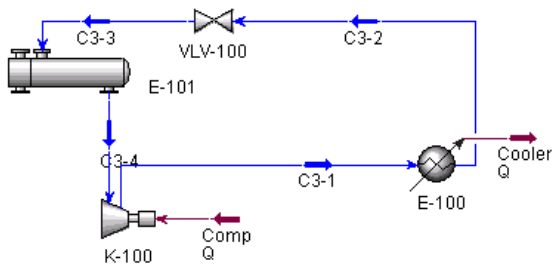
2.3.4 Example - Refrigeration Loop Template

For this example, a refrigeration loop Process Template will be constructed. This template will then be installed into a simulation case as a sub-flowsheet. The required flow of refrigerant fluid to cool a Natural Gas stream in the Main flowsheet will then be determined.

Refrigeration Loop Template

The Refrigeration Template PFD is shown in Figure 2.30:

Figure 2.30



Note that there are no material Boundary Streams in this Template Flowsheet. The **HEAT EXCHANGER** operation allows you to select a different Flowsheet for each side of the exchanger. By using this feature, the process stream does not have to be imported into the sub-Flowsheet and the Refrigeration Loop Template can be solved with only the refrigerant in its Fluid Package. The **HEAT EXCHANGER** will perform the necessary heat transfer across the Flowsheet boundary.

Flowsheet Setup

1. Create a new case.
2. For this Refrigeration Loop, the **Peng Robinson** property method will be used with components propane and propene.

Property Package	Components
Peng Robinson	C3, C3=

3. Create streams named **C3-2** and **C3-4** and define them as follows:

Name	C3-2	C3-4
Vapour Fraction	0.0000	1.0000
Temperature [C]	35.0000	<empty>
Pressure [kPa]	1742.2646	<empty>
Comp Mole Frac [Propane]	0.9500	<empty>
Comp Mole Frac [Propene]	0.0500	<empty>

4. Create streams named **C3-1** and **C3-3**.
5. Install a **COOLER** with the following specifications:

Tab [Page]	Input Area	Entry
Design [Connections]	Name	E-100
	Feed Stream	C3-1
	Product Stream	C3-2
	Energy Stream	Cooler Q
Design [Parameters]	Pressure Drop	20 kPa

6. Install a **COMPRESSOR** with the following specifications:

Tab [Page]	Input Area	Entry
Design [Connections]	Name	K-100
	Inlet	C3-4
	Outlet	C3-1
	Energy	Comp Q
Design [Parameters]	Adiabatic Efficiency	75%

7. Install a **VALVE** with the following specifications:

Tab [Page]	Input Area	Entry
Design [Connections]	Name	VLV-100
	Feed Stream	C3-2
	Product Stream	C3-3

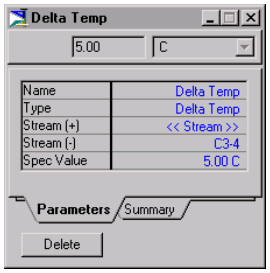
8. Add a **HEAT EXCHANGER** with the following specifications:

Tab [Page]	Input Area	Entry
Design [Connections]	Name	E-101
	Shell Side Inlet	C3-3
	Shell Side Outlet	C3-4
Design [Parameters]	Heat Exchanger Model	Weighted
	Tube Side Delta P	35 kPa
	Shell Side Delta P	5 kPa
	Pass-1 Intervals	2
	Pass-2 Intervals	2

At this point, the Tube Side Inlet and Outlet streams of E-101 will not be attached. If these streams were created now, they would, by default, inherit the components of the sub-flowsheet - propane and propene. To avoid this, the split flowsheet capabilities of the **HEAT EXCHANGER** will be used. When the Template is installed in the Main flowsheet, the process side of the exchanger will be connected directly with streams in the Main flowsheet, and will be able to use the Main flowsheet's components.

9. On the **Specs** page of the **Design** tab, add a Heat Exchanger temperature specification. Specify a **5°C temperature approach** between the **Unknown Stream** in the drop down list (select either one since this will be changed to the Tube Side Outlet stream when the Template is installed in the Main flowsheet) and **C3-4.** The specification, shown below, is accessed by pressing the ***Add*** button.

Figure 2.31



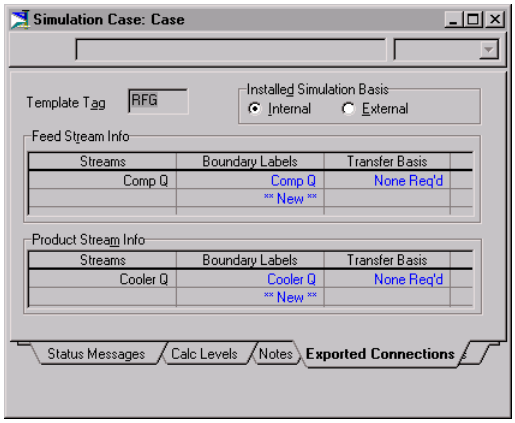
Specifying the Template Information

To open the Simulation Case property view use the hot key combination CTRL M.

1. Open the **Simulation Case** property view.
2. Press the ***Convert to Template*** button and when asked to confirm your action, select the ***Yes*** button.

3. Save the template. Provide a name **refrig.tpl**.
4. On the **Exported Connections** tab, specify a **Template Tag** of **RFG** and ensure that the ***Internal*** radio button is selected in the **Installed Simulation Basis** group.

Figure 2.32



5. The energy streams that cross the boundary should be labelled **Comp Q** and **Cooler Q**.
6. On the **Exported Variables** tab, add the following **Externally Accessible Variables**, as shown below.

*Press the **Add** button on the **Variables** to add new variables to the Externally Accessible Variables list.*

Variable Description	Object	Variable
Refrigeration Temperature	C3-4	Temperature
Condenser Temperature	C3-2	Temperature
Refrigerant Flow Rate	C3-4	Std Gas Flow
Compressor Duty	Comp Q	Heat Flow
Condenser Duty	Cooler Q	Heat Flow

7. Press the ***Save*** button to store the template.

Main Process

In this part of the illustrative example, two Natural Gas streams will be defined in the Main Simulation. The Refrigeration template, which was previously created, will be installed to cool the Natural Gas. To facilitate this, the process streams will be attached to the tube side of the heat exchanger in the template.

1. Create a new case (CTRL N).
2. Select the **Peng Robinson** property method, and add the components: methane, ethane, propane, i-butane, n-butane, i-pentane, n-pentane.

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

3. Install the stream **Cool Gas** with the following specifications:

Name	Cool Gas
Temperature [C]	60.0000
Pressure [kPa]	200.0000
Molar Flow [kgmole/hr]	500.0000
Comp Mole Frac [Methane]	0.2500
Comp Mole Frac [Ethane]	0.1250
Comp Mole Frac [Propane]	0.1250
Comp Mole Frac [i-Butane]	0.1250
Comp Mole Frac [n-Butane]	0.1250
Comp Mole Frac [i-Pentane]	0.1250
Comp Mole Frac [n-Pentane]	0.1250

4. Install the stream **Cold Gas**; specify the temperature to be -25 ^{o}C.

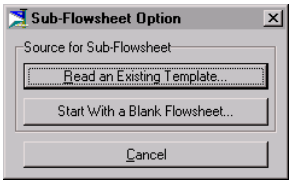
These streams are installed in the Main simulation environment and will be attached as the tube side inlet and outlet for the heat exchanger.

Installing the Template in the Main Flowsheet

***Sub-flowsheet** Button found on Object Palette.*

1. Double click on the ***Sub-Flowsheet*** button in the Object Palette.
2. From the **Sub-Flowsheet Option** dialog, press the ***Read an Existing Template*** button.

Figure 2.33



3. Select **refrig.tpl** from the **Available Sub-Flowsheet Templates** view and press the ***Open*** button.

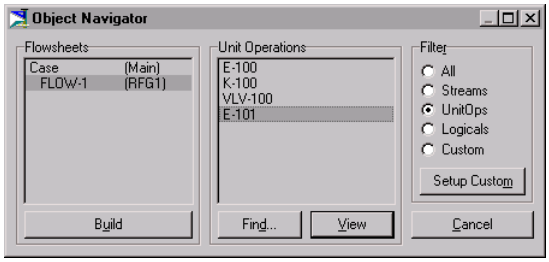
4. The sub-flowsheet property view will open to the **Connection** tab. Input the names **Comp Q** and **Cooler Q** in the appropriate **External Stream** input cells.

Connecting the Heat Exchanger Process Side

*Refer to **Section 5.2.1 - Object Navigator** for details on the Object Navigator.*

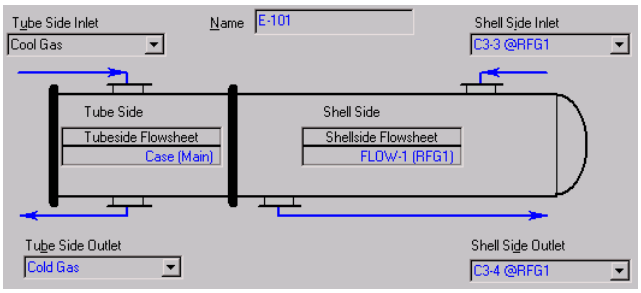
Enter the sub-flowsheet environment again and open the heat exchanger (**E-101**) view using the Object Navigator.

Figure 2.34



On the **Connections** page of the **E-101** property view, select the **Main** flowsheet as the **Tubeside Flowsheet**. Once the Tubeside flowsheet is selected, the drop down list in the Edit Bar will display Main flowsheet streams for the Tubeside connections. Select the streams **Cool Gas** for the inlet and **Cold Gas** for the outlet:

Figure 2.35

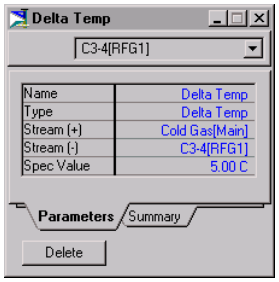


The Heat Exchanger will now solve, as will the rest of the streams in the sub-flowsheet.

1. On the **Specs** of the **E-101** property view, open the view for the temperature approach specification by highlighting it and pressing the ***View*** button.
2. Press the DELETE in the cell that displays **Cool Gas**.

3. In the Edit Bar drop down list, select the stream **Cold Gas**. The specification view should be as shown below:

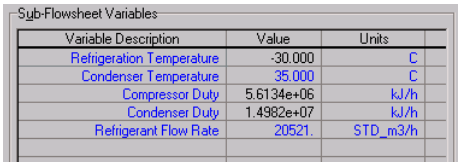
Figure 2.36



The heat exchanger will re-solve reflecting the proper approach specification.

Go to the **Parameters** page of the sub-flowsheet property view. The exported variables from the template are displayed:

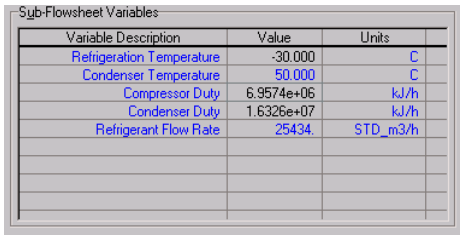
Figure 2.37



Change the **Condenser Temperature** to 50°C and observe the increased Refrigerant Flow Rate requirement and correspondingly higher duties.

*To conduct a proper case study, you can use the **Case Studies** feature in the DataBook.*

Figure 2.38



3 PFD

3.1 Object Inspection 4

3.2 PFD Tools 5

3.3 Installing Streams and Operations 9

3.4 Connecting Streams and Operations 10

3.4.1 Creating a New Stream from an Operation 11
3.4.2 Connecting Operations to Existing Streams 11
3.4.3 Connecting Two Operations 12
3.4.4 Connecting Logical Operations 13
3.4.5 Working with the Column PFD 13

3.5 Disconnecting Streams and Operations 13

3.6 The Custom PFD Notebook 15

3.6.1 Installing a New PFD 15
3.6.3 Renaming a PFD 16

3.7 Cut/Paste Functions 17

3.7.1 Creating Sub-Flowsheets 17
3.7.2 Importing/Exporting Objects 18
3.7.3 Cloning Objects 19
3.7.4 Cut/Copy/Paste 19

3.8 Flowsheet Analysis Using the PFD 19

3.8.1 Stream Label Options 21
3.8.2 Accessing Column or Sub-Flowsheet PFDs 23
3.8.3 Opening Controller Face Plates 25
3.8.4 PFD Tables 26

3.9 PFD Colour Schemes 30

3.9.1 Selecting/Changing a Colour Scheme 31
3.9.2 Adding a Query Colour Scheme 32

3.10 Manipulating the PFD 34

3.10.1 Selecting PFD Objects 34
3.10.2 Transforming Icons, Labels and Annotations 42
3.10.3 Auto-Scrolling 45
3.10.4 Rerouting 46

3.11 Adding Annotations 52

3.11.1 Editing Annotations 53

3.12 Hiding PFD Objects 54

3.13 Multi Pane PFDs 55

3.14 Printing the PFD 57

PFD Button

One of the key benefits of the PFD is that it provides the best representation of the Flowsheet as a whole. From this one location you have an immediate reference to the progress of the simulation you are currently building; i.e. what streams and operations have been installed, Flowsheet connectivity, the status of objects, etc. The PFD is HYSYS' default view and is open when you first enter the HYSYS Simulation Environment upon creating a new case.

As HYSYS' default view, the PFD will be the active window when you initially enter the Simulation Environment.

Like the other major interface elements in HYSYS, the PFD has been developed to satisfy a number of functions. In addition to graphical representation, you can build your Flowsheet within the PFD using the mouse to install and connect objects. A full set of manipulation tools has been included to allow you to reposition streams and operations, resize icons, or reroute streams. All of these tools have been designed to simplify the development of a clear and concise graphical process representation. **Figure 3.1** shows the a HYSYS case PFD and highlights some of its features.

As a key interface element in HYSYS, the PFD also possesses analytical capabilities. You can access property views for streams or operations directly from the PFD, or install custom Material Balance Tables for any or all objects. Complete Workbook pages can also be displayed on the PFD. Information will be automatically updated whenever changes are made to the process. There are several ways that you can track a specific variable throughout the PFD including the replacement of stream name labels or by designating a colour to represent a variable range.

Any PFD in the simulation can be accessed from any location using the CTRL P hot key.

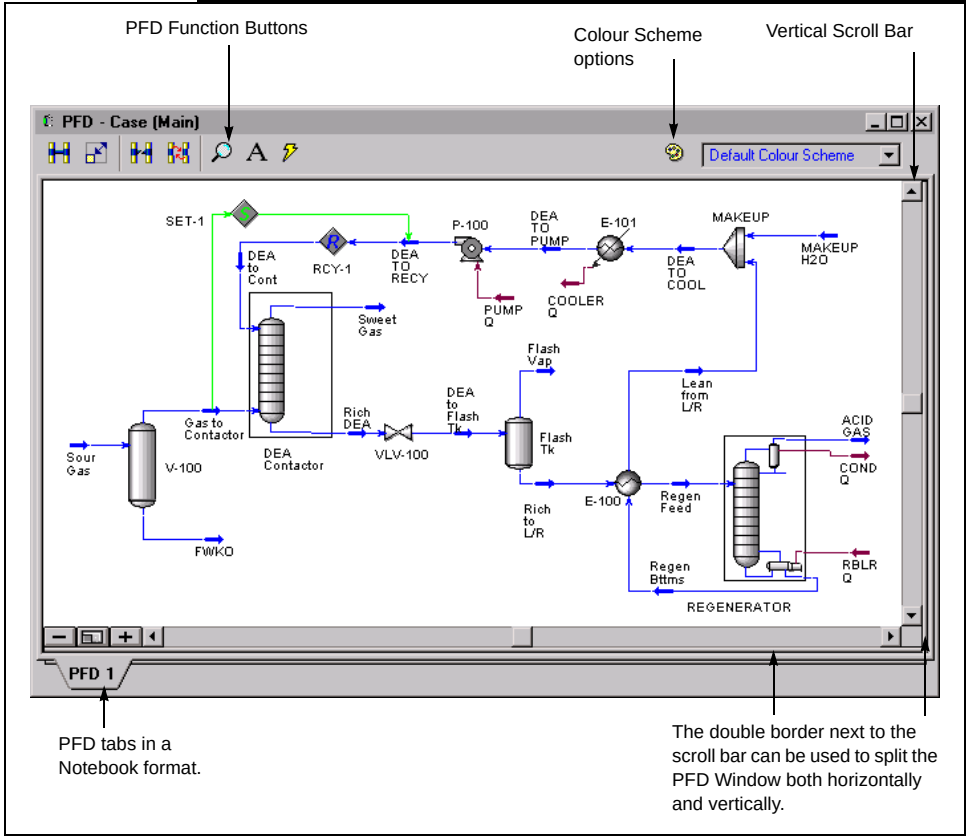
Since every Flowsheet (or Sub-Flowsheet) possesses its own PFD, and you can access any Flowsheet's PFD from any location, you can use the Multi-Flowsheeting Architecture of HYSYS to provide clear and concise representations of complex simulations. Instant access to the Sub-Flowsheet PFD is provided through object inspection of the main Flowsheet's PFD.

3.1 Object Inspection

In the PFD, there are a number of instances in which either the mouse or the keyboard can be used to perform the same function.

> One very important PFD function for which the keyboard cannot be used is Object Inspection

Figure 3.1



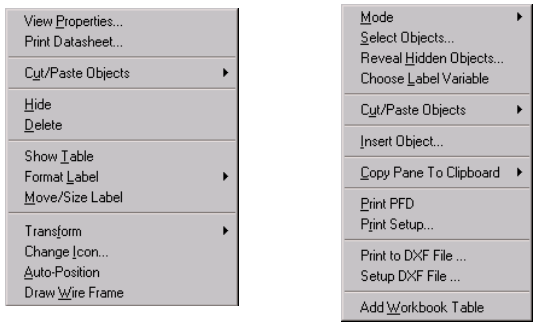
You can perform many of the tasks and manipulations on the icons in the PFD by using Object Inspection. Place the mouse pointer over the icon you want to inspect and press the secondary mouse button. An appropriate menu is produced depending upon the icon selected (Stream, Operation, Column, or Text Annotation).

The Object Inspection menu items for an Operation are shown as the first menu in Figure 3.2. **Cut/Paste Objects**, **Format Label** and **Transform**, each have an additional menu that is produced when that item is selected.

In addition to inspecting the individual icons in the PFD, you can also apply Object Inspection to the PFD itself. Press the secondary mouse button over an open area of the PFD (not on top of an individual icon). This produces a menu of available commands relating to the PFD itself, shown as the second menu to the right in Figure 3.2. **Cut/Paste Objects**, and **Copy Pane to Clipboard**, each have an additional menu that is produced when that item is selected.

*Section 3.7 - Cut/Paste Functions details the **Cut/Paste Objects** menu commands in details.*

Figure 3.2



Object Inspection Menu for **Operations**.

Object Inspection Menu for the **PFD**.

*For Sub-Flowsheet Operations (including the Column Sub-Flowsheet), the **Operations** menu also includes the following additional commands for Sub-Flowsheet manipulation:*

3.2 PFD Tools

There are several tools that help to simplify your interaction with the PFD. The most basic tools relate to what is displayed in the PFD Window (zoom level).

Pan/Zoom Functions

There will be instances when you will want to focus on a particular area of the PFD or conversely, view all objects in the Flowsheet. The following table lists functions that are available to help you display the required objects:

Function	Button	Definition
Zoom Out		Zooms display out by 25%. You can either press the ***Zoom Out*** button (minus symbol) located in the lower left of the PFD or use the keyboard command **SHIFT PAGE DOWN**.
Zoom In		Zooms display in by 25%. You can either press the ***Zoom In*** button (plus symbol) located in the lower left of the PFD or use the keyboard command **SHIFT PAGE UP**.
Zoom All		Displays all unhidden objects in the current Window. You can either press the function button located in the lower left of the PFD or use the keyboard command **HOME**.
Zoom Out display 5%		Use the keyboard command **PAGE DOWN**.
Zoom In display 5%		Use the keyboard command **PAGE UP**.
Zoom In Mouse Wheel		Click on the Mouse Wheel and scroll forward. The PFD display will zoom in.
Zoom Out Mouse Wheel		Click on the mouse wheel and scroll backward. The PFD display will zoom out.
Zoom In HOME Key	**HOME**	Select PFD objects and press the **HOME** key to zoom in on those objects. If no objects are selected, the entire PFD will be shown.
Toggle between last two Zoom views	**Z**	Use the keyboard key **Z**.
Pan 15% Left, Right, Up, Down		Use the respective keyboard arrow keys: left, right, up, down.
Pan 70% Left, Right, Up, Down		Use the **SHIFT** key combined with one of the arrow keys: left, right, up, down.
Pan Mouse Wheel		Click on the mouse wheel (or middle mouse button) and move the mouse to pan the view. To stop, click the mouse button a second time.
Centre PFD on cursor	**.**	By pressing the period key on the keyboard, **.**, the PFD will shift such that the location of the cursor becomes the centre of the view.

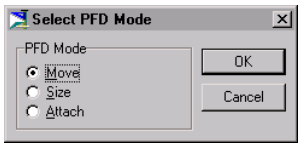
In order for the PFD to respond to the keyboard command, it must be the active window.

PFD Modes

The PFD in HYSYS operates in three modes - **Move**, **Attach**, and **Size**. Only one of these modes can be "active" at a time. This means that when you are in **Attach** mode, you cannot move or resize icons. You can switch modes via the buttons in the upper left corner of the PFD, using

Select Mode under **PFD** in the Menu Bar, or by selecting **Mode** from the PFD Object Inspection menu.

Figure 3.3



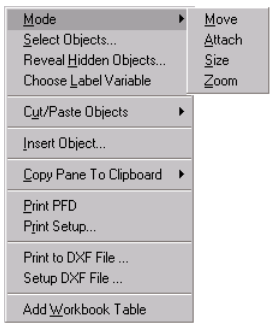
Name	Button	Definition
Move/Attach		Controls two of the PFD modes, Move and Attach. **Move**, the default setting, allows you to relocate selected operations and streams. When this button is "pressed" you are in **Attach** mode, which allows you to connect streams and operations graphically. Refer to **Section 3.4 - Connecting Streams and Operations** for more information concerning **Attach** mode.
Size Mode		When in **Size** mode, selected objects can be sized. A selected sizeable object appears with a box around it, and this box contains eight smaller white boxes around its perimeter. Using the mouse, you can drag the size of the box in any of these eight directions. Refer to **Section 3.10.1 - Selecting PFD Objects f**or more details on sizing PFD objects.

Additional Buttons

Name	Button	Function
Break Connection		When this button is selected, you can break the connection between a stream and an operation. Once you have placed the mouse pointer over an appropriate location, the cursor is redrawn with a check mark added to it. Select any portion of the stream between the stream icon and the operation. Refer to **Section 3.5 - Disconnecting Streams and Operations** for more details.
Swap Connections		Switches the nozzle connection points for two streams attached to the same operation. For more information, refer to **Section 3.10.1 - Selecting PFD Objects.**

Name	Button	Function
Drag Zoom		When you press this button, the cursor becomes an arrow and magnifying glass combination. Click and drag around a region of interest to redraw the PFD showing the selected region only. You can also Zoom from the PFD Object Inspection menu by selecting **Mode** and then **Zoom** (see **Figure 3.4**) or by selecting **Drag Zoom** under **PFD** in the Menu Bar.
Add Text Annotation		Allows you to add text to a PFD. When the button is pressed, a **'+'** symbol is added to the regular cursor and a rectangular box appears at the end of the pointer. Position the cursor where the text is to be placed, press the primary mouse button, and then type the text into the dialog box presented. Refer to **Section 3.11 - Adding Annotations**, for more information.
Colour Scheme		Displays the **PFD Colour Schemes** dialog box, from which a new scheme can be created or an existing one chosen, edited or deleted. **Refer to Section 3.9 - PFD Colour Schemes** for more detailed information.
Quick Route Mode		The **Quick Route Mode** allows icons to be quickly and independently manipulated about the PFD (i.e. icons can be moved with their attached streams overlapping the view of other icons). Upon release of this button, HYSYS repositions streams so that there is no overlap of icons. Refer to **Section 3.10.4 - Rerouting,** for more.

Figure 3.4



If you are trying to perform a function in the PFD, such as **Move**, **Size**, or **Attach**, and it is not working, check the buttons to see if you are operating in the correct mode.

3.3 Installing Streams and Operations

*See **Section 1.3.5 - Installing Operations** for a description of how to provide the necessary information to define operational parameters.*

The PFD can be used to install streams and operations into the flowsheet, as well as connect streams to operations, or operations to other operations. Object specifications are then supplied via the appropriate property view, which can be accessed by double clicking on the object icon.

The Stream and Unit Operation PFD installation procedure follows:

*For information concerning the Object Palette, refer to **Section 1.3.3 - Object Palette**.*

1. Place the mouse pointer over the desired operation in the Object Palette.
2. Hold down the secondary mouse button and drag to the desired location in the PFD. As you move the cursor, a box and target will indicate the placement of the icon.
3. Release the mouse button and the stream or operation will be installed.

or

Streams and Operations added via any other method automatically appear in the PFD with connections already drawn.

1. From the Palette, push the button for the desired operation.
2. Click (with the primary mouse button) on the location of the PFD where the new object is to be placed.

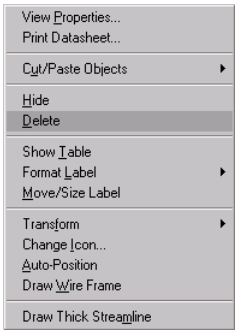
You can move the icon to any open space on the PFD.

Deleting Streams and Operations

Streams and operations can be deleted from the case using the PFD in any of the following ways:

- Select the object you wish to delete and press the **DELETE** key on the keyboard.
- Object Inspect the appropriate icon and choose Delete from the menu, as shown in **Figure 3.5** below:

Figure 3.5



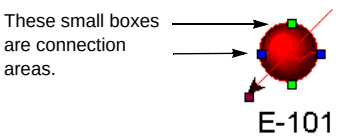
With either method, HYSYS prompts you to confirm deleting the object. If you select multiple objects to delete, HYSYS will ask for confirmation before deleting each object.

3.4 Connecting Streams and Operations

Connection nozzles are the small coloured connection points that appear on streams and operations when the cursor is passed over top of them when the PFD is in ***Attach*** *mode.*

You can connect an operation to an existing stream or have a new stream created to connect two operations. New streams can be created by dragging from an existing operation connection nozzle in **Attach** mode.

Figure 3.6



Connection Nozzle Colours:
Red - Energy Stream
Blue - Material Stream
Green - Logical Connection

When the PFD is in **Attach** mode, connection nozzles will automatically light up for each icon as the mouse pointer passes over the icon. A red box indicates an Energy Stream connection, a blue box indicates a Material Stream connection, and a green box indicates a Logical connection. An exception to this is the **BALANCE** block which can accept both Material and Energy Streams. All connections for the **BALANCE** are shown in green.

In addition to the coloured squares which indicate the different connection types, a fly-by description appears when the mouse pointer is placed over a connection point.

3.4.1 Creating a New Stream from an Operation

*Hold down the CTRL key to temporarily toggle to **Attach** Mode. Make the connections and release the CTRL key.*

In order to use the PFD to create a stream, the ***Attach*** button in the upper left corner of the PFD must be pressed.

The procedure for creating a new stream is as follows:

1. Place the cursor over the desired operation connection point, or nozzle. When the cursor is in the correct location, the perimeter of a white square will appear at the tip of the mouse pointer.
2. Hold down the primary mouse button, and begin dragging the stream to an empty space in the PFD. First a full black square appears at the end of the mouse pointer. Keep dragging the stream until the Stream Connection Tool (as shown to the left) appears at the end of the stream. If you release the mouse button while the full black square is shown, the stream will not be attached.
3. Release the mouse button. HYSYS will place a stream icon here and name it using the next stream name as defined under the current Preferences.

Stream Connection Tool

*See **Section 1.3.4 - Installing Streams** for a description of how to provide the necessary information to define the stream conditions and compositions.*

3.4.2 Connecting Operations to Existing Streams

Move / Attach Button

You can either connect a stream to an operation, or an operation to a stream. The procedure for both is identical. Once again, to perform any of these operations, you must press the ***Move/Attach*** button to switch into **Attach** mode.

*See **Section 3.10.4 - Rerouting** for information on manually changing the route of a stream.*

To connect an operation to an existing stream, place the pointer over the desired nozzle, hold down the primary mouse button and begin dragging to the desired stream. As you move the mouse, HYSYS draws a line indicating the creation of a stream. This line tracks the mouse directly. You cannot prescribe a path for the stream to follow as you perform the connection.

> Note that both Streams and Operations have Inlet and Outlet nozzles. The connection points that light up depend on the origin of the connection, i.e., when connecting from a Stream Outlet, only Inlet nozzles to operations will light up.

Stream Connection Tool

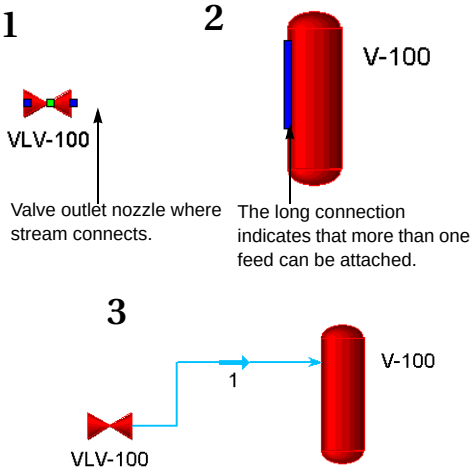
As you approach the stream, the available connection will light up. When you are within the defined connection region for a nozzle (larger than the region used when HYSYS indicates available connection points), the cursor will change to the **Stream Connection Tool**. To complete the connection, approach the appropriate nozzle and release the mouse button when a full white square appears at the tip of the mouse pointer.

3.4.3 Connecting Two Operations

Connecting two operations directly will automatically create a new stream (using the next available name as defined in the **Session Preferences**). The procedure is as follows:

1. Press the ***Move/Attach*** button or hold down the CTRL key.
2. Select the nozzle of the operation from which the stream will be connected (in the case of Figure 3.7 the Valve, **VLV-100**).
3. While holding down the primary mouse button, drag the new stream toward the operation to which it will be connected, the separator **V-100** in Figure 3.7. HYSYS indicates the available connection points. In this case, because the stream is being taken from the outlet of the Valve, the only connection point indicated is the inlet area of the Separator.
4. When you reach the connection region, the mouse pointer changes to a solid white square. Release the mouse button at this time to complete the connection and create a new stream.

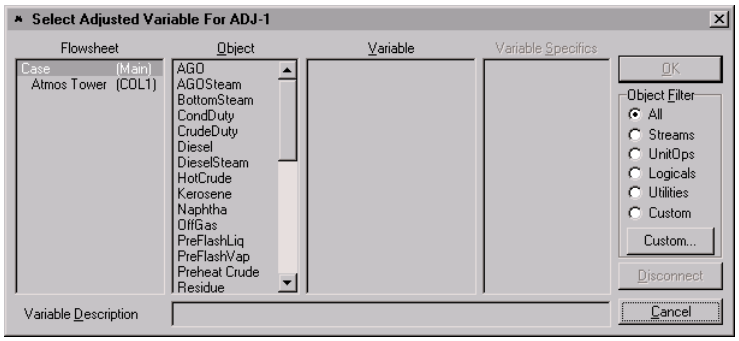
Figure 3.7



3.4.4 Connecting Logical Operations

Logical Operations can be connected to Operations or Streams in the same way as explained previously. The only difference is that once the connection is made, a dialog box appears. A variable, specific to the object to which the logical is connected, is chosen. This view is different depending on the type of logical operation and whether the connection has been made to a stream or operation. Consider the situation where a connection from an **ADJUST** operation to a Stream is made; the stream is the Adjusted Variable Object. Figure 3.8 shows you the HYSYS prompt you use to enter the Variable for the Adjusted Object.

Figure 3.8



3.4.5 Working with the Column PFD

When you are inside the **Column Sub-Flowsheet**, you can connect Feed and Draw streams to the Column Tray Section in the PFD. The Column PFD shows all column stages. Connections can be made to each stage. The procedure for connecting Streams to the Column is the same as that for Connecting Streams to Operations.

3.5 Disconnecting Streams and Operations

Sometimes it is necessary to break connections between streams and operations. There are several ways to accomplish this task from the PFD, without having to enter the operation's property view.

Using Object Inspection

You can Object Inspect a connection line to break the connection:

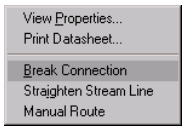
1. Move the pointer to the stream where you want to break the connection.
2. When the pointer is on the stream away from the arrow icon, it will change into the Move Segment cursor (for a horizontal line segment, the cursor is a vertical line with arrowheads at each end, as in Figure 3.9).

Figure 3.9

3. Press the secondary mouse button. From the menu that appears, choose **Break Connection**. Depending on the selected stream, the object inspection menu may not have all of the options as shown in Figure 3.10.

*The **Swap Attachments** option will appear in the object inspection menu only when more than one stream is attached to a certain location (i.e. 2 feed streams).*

Figure 3.10



Use of the Break Connection Option

Break Connection Button

The **Break Connection** option allows you to break an existing stream connection and can be accessed either by selecting the ***Break Connection*** button or selecting the PFD Menu Bar Option.

The ***Break Connection*** option (button or menu item) is available only when the PFD is open. You can only break one connection at a time. If you want to break a second connection, you will need to select the **Break Connection** option again.

Remember that breaking the connection does not delete the stream, but only its connection to the operation. You can either break an inlet or an outlet stream connection, depending on which side of the stream icon you select.

Break Connection Procedure

Once the **Break Connection** option has been selected, the following procedure must be followed:

1. Select the **Break Connection** option (via button bar or drop down

menu), then move the cursor to the stream where you want to break the connection.

2. When the cursor is on the appropriate spot, it will change from an **X** to a check mark.
3. When the check mark appears, press the primary mouse button.

3.6 The Custom PFD Notebook

Deleting an object is a global function, so if an object is deleted from a PFD, it disappears not only from all PFDs but from the simulation case.

Similar to the design of the Workbook, the PFD can be customized into a Notebook through the addition of new PFD tabs. The availability of multiple tabs in a PFD format provides greater flexibility for the user. For very large and complex process flow diagrams, one PFD can be used as the main tab that shows the whole process, while subsequent tabs show specific areas of interest. Another ideal circumstance for the PFD Notebook occurs when you want each PFD to have a distinct colour scheme for an identical objects setup.

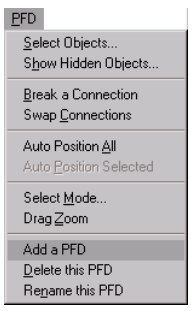
> Each PFD is an independent element on which objects can be moved or hidden without changing the appearance of the other tabs. However, when multiple PFDs exist, an object added to one PFD is shown on all PFDs.

3.6.1 Installing a New PFD

To add a new PFD to the Notebook:

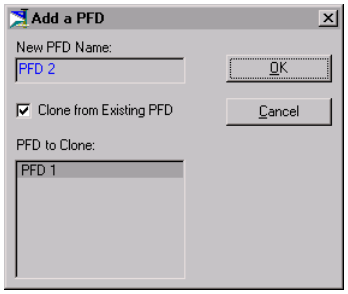
1. Select **PFD** from the Menu Bar.
2. From the drop down menu, choose **Add a PFD**.

Figure 3.11



3. The **Add a PFD** dialog box will appear, in which the option to clone an existing PFD is presented. Cloning a PFD produces an exact duplicate of the chosen PFD. To clone a PFD, activate the **Clone from Existing PFD** check box. You can change the name of the PFD in the **New PFD Name** cell. Press ***OK***.

Figure 3.12



4. A new tab appears with the specified name and the new PFD becomes the active view.

3.6.2 Deleting a PFD

There is no way to recover a deleted PFD.

Unlike the deletion of a single object, deleting a PFD will remove it from the notebook, but will not remove the associated objects from the simulation case. The procedure for deleting a PFD is as follows:

The delete option is not available if there is only one PFD in the notebook.

1. Make the PFD active by selecting its page tab.
2. In the Menu Bar, choose **PFD**.
3. Select **Delete this PFD**.

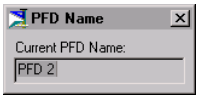
3.6.3 Renaming a PFD

If the name of the PFD was not changed when it was added, or if the name of the original PFD is not suitable, it can be easily modified.

1. Select the tab of the PFD you want to rename.
2. In the Menu Bar, choose **PFD** to access the drop down menu.

3. Select **Rename this PFD** to make the **PFD Name** dialog box appear.

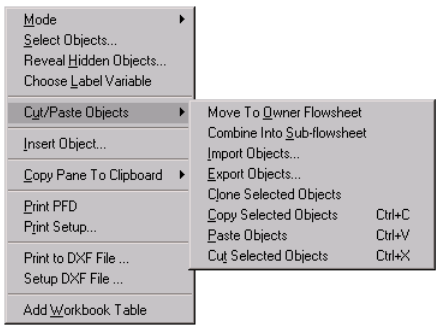
Figure 3.13



4. Modify the PFD name in the **Current PFD Name** input cell and press the ***Close*** button.
5. The new name appears on the tab.

3.7 Cut/Paste Functions

Figure 3.14



The **Cut/Paste Objects** sub-menu available in the PFD object inspection menu provides several options for adding, removing or recombining flowsheet objects.

3.7.1 Creating Sub-Flowsheets

The **Combine Into Sub-flowsheet** command allows you to select any number of PFD objects already installed in your simulation case in the main flowsheet and create a sub-flowsheet containing those objects. This feature is especially useful if you wish to organize complicated flowsheets. For instance, you may wish to divide your flowsheet into different sections in order to aid reading for clearer understanding the case. To create a sub-flowsheet in this manner:

1. Select the PFD objects that are to be part of the new sub-flowsheet. This may include unit operations, streams and logical operators.
2. Object inspect an open area of the PFD.
3. From the **Cut/Paste Objects** sub-menu, select **Combine Into Sub-flowsheet**.

HYSYS will create a new sub-flowsheet within your case.

Moving Sub-Flowsheet Objects Back to Parent Flowsheet

If you want to move the contents back to the owner flowsheet, object inspect the sub-flowsheet icon and select **Move Contents to Owner Flowsheet**.

> Note that even though the contents of the sub-flowsheet have been moved to the owner flowsheet, the sub-flowsheet still exists. You may wish to delete this empty flowsheet from your case. If you decided to "re-collapse" the sub-flowsheet objects, HYSYS will create a new sub-flowsheet.

3.7.2 Importing/Exporting Objects

You can export flowsheet objects from one and import to another using the **Import Objects** and **Export Objects** commands on the PFD object inspection menu. Exported object files have the extension .hfl.

Exporting Objects

Select the PFD objects you wish to export and then select the **Export Objects** command. HYSYS will ask you to provide a name and destination for the information.

When you export objects from a flowsheet, the objects, connections and geometric data is exported. None of the basis or flow information (components, flowrates, etc.) is included.

Importing Objects

Templates can be imported into a flowsheet by using the **Import Objects** command on the PFD object inspection menu. All basis information already supplied to the flowsheet will automatically be

applied to the imported objects. None of the basis information from the case from which the objects were exported has been saved in the template file.

3.7.3 Cloning Objects

Clone flowsheet objects on your PFD using the **Clone Selected Objects** from the Copy/Paste Objects sub-menu. Cloned objects occurs within a single flowsheet. All object information will be automatically cloned into a new set of objects, the only change will be to the object name.

3.7.4 Cut/Copy/Paste

Hot Keys:
Cut - CTRL X
Copy - CTRL C
Paste - CTRL V

The **Cut**, **Copy** and **Paste** commands in the **Cut/Paste Objects** sub-menu have the typical functionality associated with these commands. Select objects in one flowsheet and paste the cut or copied objects to sub-flowsheets within the same case. It is important to note that if the destination sub-flowsheet was created using a different fluid package, some of the copied information may not be transferrable and thus omitted.

3.8 Flowsheet Analysis Using the PFD

Once you have installed your streams and operations in the PFD, you will need to supply specifications. Specifications are generally input through the Workbook or an object's property view. Property views can be accessed through the PFD allowing you to keep the PFD open while supplying the necessary object information. In addition, Tables containing specified variables for streams and operations can also be installed on the PFD.

*If you are in **Move** or **Size** mode, a box will appear around the operation when you select it.*

*The V or E key on the keyboard can be used to open a **selected** object's property view.*

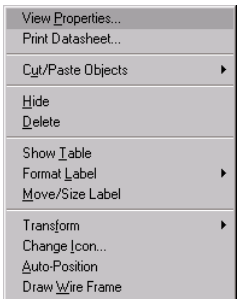
Accessing Stream and Operation Property Views

You can open property views directly from the PFD by double clicking on the icon of the desired item.

You can also use Object Inspection to access a property view. Click on an icon with the secondary mouse button. From the Object Inspection menu, select the **View Properties** option and HYSYS displays the

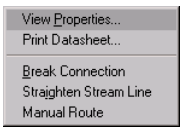
Property View for the stream or operation. The object inspection menu for a unit operation is shown in Figure 3.15.

Figure 3.15



To access the property view for a stream, you do not need to find its icon. You can double click or object inspect any portion of the line which represents the stream. By object inspecting a line segment, you will access a menu from which you can select **View Properties**.

Figure 3.16



*Refer to **Section 5.4 - Object Status Window/Trace Window** for details concerning the Trace Window.*

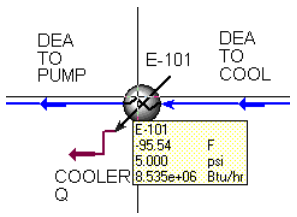
Using the Object Status Window is an equally effective method for opening property views. By moving the mouse pointer to the extra thick border (directly above the Status Bar) at the bottom of the DeskTop, the cursor will change to a vertical line with two arrowheads. Click the primary mouse button and drag the cursor upward to expose the Object Status and Trace Windows. The left pane is the Object Status Window and contains object status messages for the various streams and operations. By double clicking on a message, the property view for the associated object will appear.

Note that the Object Status Window option will not be available once the case has solved. At this point, all object status messages are **OK** and are no longer displayed in the Window.

Fly-by Information Boxes

Information related to an object can be displayed by simply placing the mouse pointer over its associated icon. HYSYS automatically shows a small box listing the object name and the current values of key variables. This is shown in Figure 3.17, displaying the fly-by description of a cooler.

Figure 3.17



3.8.1 Stream Label Options

By default, each stream on the PFD has a label that displays its name. You can change all stream name labels so that the current value of a key variable is shown in place of each stream name.

Common Variable Choices

There are a few hot key combinations that allow you to toggle between stream name labels and some of the more common stream variables:

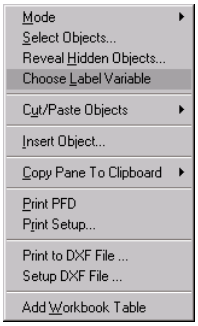
Hot Key Combination	Function
SHIFT T	Displays stream temperatures
SHIFT P	Displays stream pressures
SHIFT F	Displays stream molar flowrates
SHIFT N	Displays stream names

Other Variables

You can also replace the stream name labels with other variable values. HYSYS provides an extensive list of possible choices. As an example, a procedure for changing the stream name label to the stream Std Gas Flow value is be outlined:

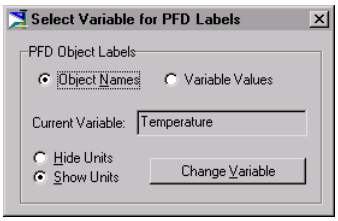
1. Object Inspect an open area of the PFD.
2. From the object inspection menu, select **Choose Label Variable**.

Figure 3.18



3. The **Select Variable for PFD Labels** dialog will appear.

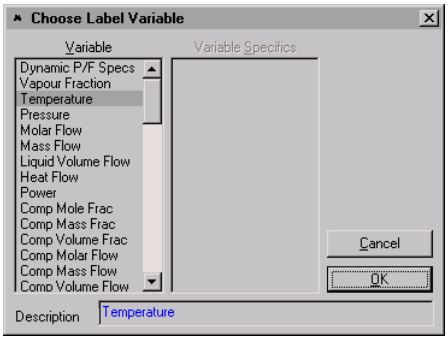
Figure 3.19



4. Press the ***Change Variable*** button.

5. From the **Choose Label Variable** dialog, select **Std Gas Flow** in the **Variable** group.

Figure 3.20

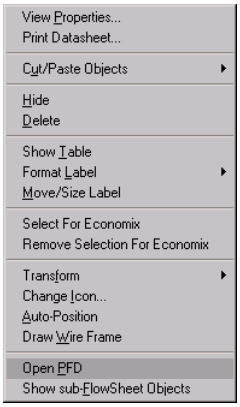


6. To modify the name of the variable, make changes in the **Description** input cell.
7. Press ***OK***.
8. You will now be returned to the **Select Variable for PFD Labels** dialog. Notice that the description provided on the previous dialog appears in the **Current Variable** cell.
9. If you would like to show the molar density of each stream without units, select the **Hide Units** radio button.
10. If you wish, you can return the stream names to the PFD by selecting the **Object Names** radio button.
11. Press the ***Close*** button.

3.8.2 Accessing Column or Sub-Flowsheet PFDs

Object Inspecting a Column (or Sub-Flowsheet template) in the Main PFD provides a menu similar to the one for streams and operations with the addition of two options: **Open PFD** and **Show sub-FlowSheet Objects**.

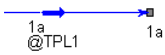
Figure 3.21



*A reasonable arrangement of icons on the PFD should be preserved using the **Show sub-FlowSheets** option, however, for complex sub-flowsheets the icon layout may become crowded and may need to be re-arranged manually.*

The **Show sub-FlowSheet Objects** command displays all sub-flowsheet objects (streams, unit operations, tables, text etc.) onto the Main Flowsheet PFD. In this way, all sub-flowsheet streams and operations can now be viewed and accessed from the main PFD without entering the sub-flowsheet environment.

Appearance of sub-flowsheet stream tip within Main flowsheet PFD.



To hide sub-flowsheet objects displayed in the main PFD, object inspect the tip of any sub-flowsheet stream connected to the main flowsheet. The tip of the stream will have a small square visible. Upon object inspection, the **Hide sub-FlowSheet Objects** option will appear which, once selected will hide all associated sub-flowsheet objects.

Column Sub-Flowsheet

In HYSYS, the external view of the Column resides in the Main PFD. Only the external streams of the Column appear, i.e. Inlet, Outlet and Energy streams. However, the Column possesses its own unique PFD, which displays the complete representation of the column Flowsheet internal view (including reboilers and condensers).

For the case of a Column, the **Open PFD** command opens the PFD, showing complete column information. The Column PFD displays the Columns internal streams, such as **Boilup** and **Reflux**. Also, in the Column PFD, the Column is shown with the appropriate number of stages.

When the column PFD is accessed from the Main Environment, you will not be able to modify internal sub-flowsheet connections. You must enter the Column Sub-Flowsheet environment to perform such tasks as adding or deleting objects and breaking stream connections.

3.8.3 Opening Controller Face Plates

Like streams, operations, and columns, you can also object inspect controllers. The options available are the same as those associated with the other operations, with the addition of one, **Face Plate**. Selecting this option will open the face plate of the object inspected controller.

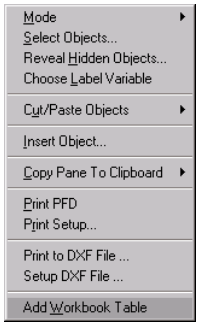
Figure 3.22

3.8.4 PFD Tables

Workbook Table

You can display any full Workbook tab as a table on the PFD, with the exception of the default **Unit Ops** tab. All information displayed on the Workbook tab will be shown in the table and will be automatically updated when changes occur in the Flowsheet.

Figure 3.23



Place a Workbook table on the PFD by object inspecting any open area of the PFD and selecting **Add Workbook Table**. The **Select Workbook Page** dialog will appear from which you can highlight any of the existing Workbook tabs and press ***Select***.

Once the Workbook table is placed on the PFD, you can click and drag it to a new location.

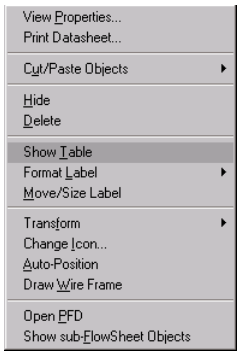
When you object inspect a Workbook table, the menu shows the following options:

Menu Item	Description
Hide	Hides the Workbook table.
Change Font	Allows you to change the font for the text in the table
Change Colour	Opens the colour palette, so you can change the colour of the table text, and the table outline

Object Variable Table

You can install a Table for any object appearing in the PFD by object inspecting the stream or operation and selecting **Show Table** from the **Object Inspection** menu.

Figure 3.24



Each Object Type has a default variable list associated with it. The Table shown here is for the operation **Flash Tk**. You can modify the list of displayed variables, as well as the table appearance.

Figure 3.25

Flash Tk		
Separator Type	Separator	
Vessel Temperature	134.7	F
Vessel Pressure	90.00	psia
Liquid Molar Flow	4295.	lbmole/hr
Duty	0.0000	Btu/hr

When you object inspect a Table, the menu shows the following options:

Figure 3.26

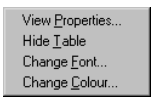


Table Menu Item	Description
View Properties	Accesses the various properties available for the tables. Refer to the **Table Properties** section for a detailed explanation.
Hide Table	Temporarily hides the table on the PFD. When the table is revealed, it will be shown as it was before it was hidden.

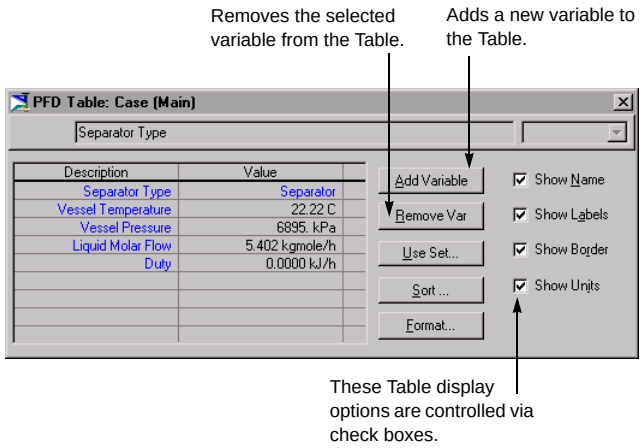
Table Menu Item	Description
Change Font	Allows you to change the font for the text in the table.
Change Colour	Opens the colour palette, so you can change the colour of the Table text, and the Table outline.

Table Properties

*You can also bring up the **PFD table** dialog by double clicking on the table.*

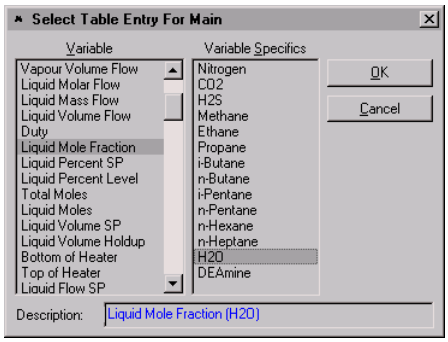
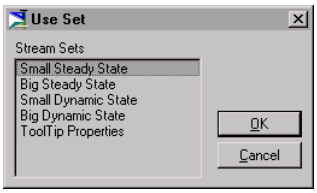
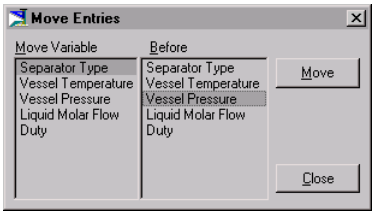
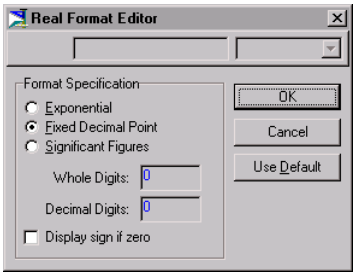
A default variable set is provided for each stream and operation type. To modify this list, object inspect the Table and select the **View Properties** option.

Figure 3.27



*If you are having trouble selecting a table, make sure that you are not in **Attach** mode.*

The buttons located within the dialog box can be used to modify the variable set for the table. Descriptions of the function and use of the buttons follow:

Button	Button Usage	Views / Remarks
Add Variable	This button allows you to add a variable to the list of variables appearing in the Table. The procedure is as follows: 1. Select the ***Add Variable*** button to open the **Select Table Entry for Main** view. 2. From the **Variable** column, highlight the variable you wish to add to the Table. 3. If you choose a variable such as **Liquid Mole Fraction**, the available components appear in the **Variable Specifics** column. Select a variable specification, in this case **H2O.** 4. You can change the variable description in the **Description** box at the bottom. 5. Press the ***OK*** button to return to the **PFD Table** dialog. Note that the **Add Variable** option allows you to add only one variable at a time to the Table.	
Remove Var	This button lets you remove variables from the table. Select the variable(s) to remove from the **Description** column, and press the ***Remove Var*** button.	*HYSYS does not ask for confirmation before deleting variables from the table.*
Use Set	HYSYS typically provides default variable sets, accessed via the ***Use Set*** button. The list of variable Sets differs, depending on the object type (stream, unit operation, column, controller). To change to a default variable set, select one from the list and press ***OK***. The sets shown to the side apply to a Material Stream Table.	
Sort	This button allows you to reorganize variables in the table. Select the item(s) you wish to move from the **Move Variable** column. In the **Before** column, select the variable in front of which you want the previously selected item(s) to move. Press the ***Move*** button. Movement of single as well as groups of variables is allowed. To group consecutive variables, hold down the **SHIFT** key while making selections. For non-consecutive variables, hold down the **CTRL** key.	
Format	With this button you are able to change the numeric format of a table value. Before pressing the ***Format*** button, select the variables you want to change on the Properties table. In the **Real Format Editor**, in the **Format Specification** group box you can choose from the radio buttons available: Exponential, Fixed Decimal Point or Significant Figures. Below the radio buttons you should further specify your selection. Press ***OK*** to change the table value.	

Column Tables

Column tables can be added in the Main PFD and in the Column PFD. The tables that can be added are different in each environment. For instance, in the Main PFD the Column Table consists of variables relating to the Column Tray Section.

Inside the Column PFD you can add Tables for the Condenser, Tray Section, and Reboiler. Each of these can contain variables specific to that unit operation.

3.9 PFD Colour Schemes

Colour Scheme Button

Each PFD can have its own distinct Colour Scheme.

By simply viewing the colour schemes on the PFD, you can retrieve specific information about your case. The type of information that is available depends on the selected colour scheme. For example, in **Default** mode, a unit operation may be red, which indicates a serious status message associated with the object. The colour red may indicate that the object requires the attachment of a material or energy stream. The benefit of the colour scheme in the PFD is greatly enhanced if the object status window is also open. The object colour combined with the information provided in the object status window can prove very helpful.

There are three colour scheme options:

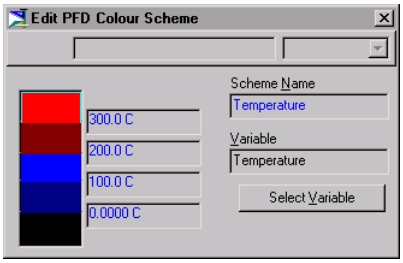
Scheme	Description
Default Colour Scheme	This colour scheme changes the colour of unit operations and streams to reflect the status of the object. When building a flowsheet from the PFD, use of this colour scheme is advantageous. Unit Ops are **red** if there is a serious message in the status window, outlined in **yellow** if a warning message exists and completely **grey** if the object has successfully solved. A Stream icon changes colour once its status message shows **OK**. The default colours are light blue for unsolved and dark blue for **OK**, but can be changed in the Session Preferences.
HYSIM Colour Scheme	Streams and unit operation icons are shown as wire frames. You may change the colour of the wire frame. Object inspect an object choosing **Change Colour** from the menu. The colour palette appears and a new colour can then be selected. Choose an existing colour or press the ***Define Custom Colours*** button to customize a colour. Once a colour has been chosen, press the ***OK*** button. The new colour for the wire frame will then appear.

*PFD default colours can be changed on the **Colours** page of the Session Preferences view. Refer to **Section 7.7.5 - Preferences**.*

Simultaneously change the colour of multiple wire frames by selecting all of the desired icons.

Scheme	Description
Query Colour Scheme	The value of a specified variable can be monitored for all material streams. You can choose five colours and an associated variable range for each, as shown to the side, for stream pressure. For the example given in **Figure 3.28**, the top colour (Colour 1) will appear for material streams that have a temperature greater or equal to 300C. Colour 2 represents streams ranging from 200 to 300C. Colours 3 and 4 are explained similarly. The last colour (Colour 5) is shown for streams that have temperatures below 0.000C. Refer to the following sections for information on working with query colour schemes.

Figure 3.28



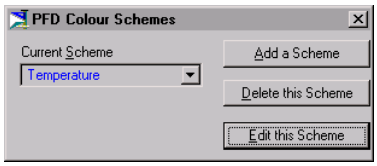
3.9.1 Selecting/Changing a Colour Scheme

The Default Colour Scheme is active when the PFD is first accessed. There are two ways of switching to another scheme:

- Click on the ***Colour Scheme*** button in the button bar. This produces the PFD Colour Schemes dialog box (**Figure 3.29**). Open the **Current Scheme** drop down list and select a colour scheme. Press the ***Close*** button.

Colour Scheme Button

Figure 3.29



- Use the mouse or the **TAB** key to open the colour scheme drop down menu, located to the far right of the button bar. Select a colour scheme from the list.

When the simulation case is saved, the active colour scheme for each PFD is also stored. Note that the ***Delete this Scheme*** and ***Edit this Scheme*** buttons appear for **Query Colour Schemes** only.

3.9.2 Adding a Query Colour Scheme

To add a colour scheme that will track a key material stream variable throughout the PFD, follow this procedure:

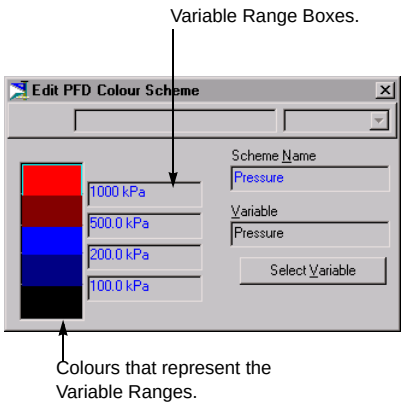
*The Temperature colour scheme, which is shown in the **Colour Scheme** drop down list when the PFD is accessed, is a Query scheme.*

1. Click on the ***Colour Scheme*** button in the button bar.
2. Press the ***Add a Scheme*** button in the **PFD Colour Schemes** dialog box. The Variable Navigator appears.
3. Choose a material stream variable from the **Variable** group box and if appropriate, select the variable specifics (i.e. for the selection of Comp Mole Frac, a component must be chosen from the **Variable Specifics** group box).

Refer to Section 5.2.2 - Variable Navigator for more details.

4. Press the ***OK*** button. The **Edit PFD Colour Scheme** dialog box appears.
5. Input four values in the variable range boxes.

Figure 3.30



*For details on available colour scheme changes, refer to the section **Editing a Query Colour Scheme**.*

6. Double click on a colour to access the colour palette, from which you may select a different colour for a variable range. Changes can also be made to the name of the **Scheme Name** and to the **Variable**.

7. Select the ***Close*** button. This returns you to the **PFD Colour Schemes** dialog box.
8. Press the ***Close*** button.

Deleting a Query Colour Scheme

Make sure you select the right scheme when deleting as there is no confirmation required for this procedure.

__Edit__ and __Delete__ buttons appear only when a Query Colour Scheme is selected.

To delete a query colour scheme:

1. Click on the ***Colour Scheme*** button in the button bar.
2. Select a colour scheme from the **Current Scheme** drop down list in the **PFD Colour Schemes** dialog box.
3. Press the ***Delete this Scheme*** button.

Editing a Query Colour Scheme

Proceed as follows to edit a query colour scheme:

1. Click on the ***Colour Scheme*** button in the button bar.
2. Select a colour scheme from the **Current Scheme** drop down list in the **PFD Colour Schemes** dialog box.
3. Press the ***Edit this Scheme*** button.
4. On the **Edit PFD Colour Schemes** dialog box which appears, you can edit the following:

Variable	Description
Scheme Name	Use the hot key combination **ALT N** or the mouse to select the **Scheme Name** input box and enter a new name. Note that the Scheme Name will become the Query Variable name if the Variable is changed.
Query Variable	Press the ***Select Variable*** button to access the **Select Query Variable** dialog box. Choose a material stream variable from the **Variable** group box and if appropriate, select the variable specifics (i.e. for the selection of Comp Mole Frac, a component must be chosen from the **Variable Specifics** group box).
Variable Ranges	Move the cursor to one of the input boxes and input a new value. The input values will represent the limits for the five variable ranges. The **TAB** key can be used to move from the top cell to the bottom cell, while the **SHIFT TAB** is used to move in the opposite direction.

Only one colour can be changed at a time.

Variable	Description
Colours for Variable Ranges	Double click on the colour you want to change. The colour palette will appear from which a new colour can be selected. Choose an existing colour or press the ***Define Custom Colours*** button to customize a colour. Once a colour has been chosen, press the ***OK*** button and the new colour will appear in the **Edit PFD Colour Scheme** dialog box.

5. Once all changes have been made to the query colour scheme, press the ***Close*** button.
6. Press the ***Close*** button on the **PFD Colour Schemes** dialog.

3.10 Manipulating the PFD

A Show/Hide option also exists for displaying sub-flowsheet objects on the Main flowsheet PFD. For further details, see ***Section 3.8.2 - Accessing Column or Sub-Flowsheet PFDs****.*

There are a number of features built into the PFD interface that allow its appearance to be modified. The manipulations apply to all objects that can be installed in the PFD: Streams, Unit Operations, Logical Operations, Labels, and Text Annotations. In addition to the selection of alternate icons for the operations, you can manipulate the routing of streams, swap nozzle connections for two streams attached to the same operation, move and size icons, add text, transform the orientation of objects, and change text fonts and colours.

You can modify the appearance of the PFD using a number of different tools: the Menu Bar, Mouse, Keyboard, Object Inspection Menu, and PFD Inspection Menu. Each tool can be used in some manner to alter the appearance of the PFD. For whichever route you use, manipulation of the PFD works through the selection of items.

3.10.1 Selecting PFD Objects

When an object has been selected, its icon is surrounded by a white border and the label background is highlighted.

Before you can manipulate the PFD, you must know how to select PFD objects, (streams, operations, and text annotations). HYSYS allows selection of single objects as well as multiple objects, but in order to select an object, you must **not** be in **Attach** mode.

Single Object Selection

Press and hold SHIFT *to blink the bounding rectangles of items selected in the PFD*

To select a label, Object Inspect the label or the object to which it is attached, and choose ***Move/Size Label****. Alternatively, select the object and press the* L *key.*

- Position the mouse pointer on top of the object. Click once with the primary mouse button. The selected object appears with a white rectangular box around its Icon.
- You can also use the keyboard: **S** to cycle through all items in the PFD, **SHIFT S** to cycle backwards through all items.

Multiple Object Selection

There are three methods available to selecting multiple objects.

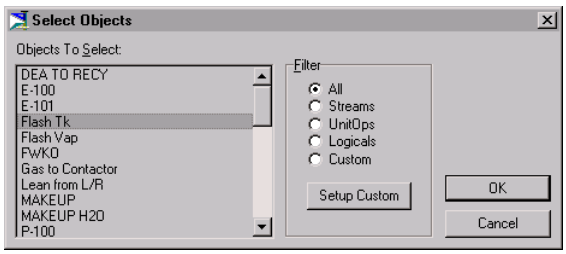
Mouse Drag Option:

1. If the objects are all contained within the same area, the quickest and easiest way is to drag a box around the group. Press the primary mouse button (outside the group), and drag the mouse so that a box appears.
2. Continue dragging until this box contains all the objects you want selected.
3. When you release the mouse button, each object will have its own outline surrounding it, indicating it has been selected.

Menu Bar Option:

1. Choose the **Select Objects** option, either in the Menu Bar under **PFD**, or from the PFD Inspection menu. This produces the **Select Objects** dialog.
2. From the **Objects to Select** group, select all of the desired objects.

Figure 3.31



Use the Filter for a more specific list in the Objects to Select.

3. When selecting multiple objects from the **Objects to Select** group, use the SHIFT key or click and drag with the mouse, provided the selections are consecutive. You can highlight non-consecutive selections with the CTRL key and the primary mouse button.
4. Press the ***OK*** button when you have made all your selections. When you close this view and return to the PFD, all of the objects you selected will be surrounded by a flashing white box.

Keyboard/Mouse Option

1. Position the mouse pointer (in the PFD) on the first object you want to select.
2. Press the primary mouse button to select this object.
3. To select a second object, hold down the SHIFT key, and click on the second object with the primary mouse button. Two objects are now selected (indicated by a white outline around each).
4. Continue depressing the SHIFT key until you have selected all the desired objects.

Deselecting Items

Any of the following methods can be used to deselect an object:

- Click on an empty spot in the PFD with the primary mouse button.
- Press the D key to deselect all the selected items.
- To unselect only one item, press the SHIFT key and click on the object with the primary mouse button.

Moving Objects

*Objects can be moved in either **Size** or **Move** mode.*

You can move objects individually, or as a group.

1. Select the item or items you want to move.
2. Position the mouse pointer over one of the selected items. Press and hold the primary mouse button.
3. Drag the mouse to the new position on the PFD and release the mouse button. All selected items will move.

*The internal **Auto Snap** feature may prevent an icon from being moved with the keyboard arrow keys. See the Section - Aligning Icons for more.*

You can also use the keyboard to move an item within the PFD. Select the item(s) you wish to move. Use the arrow keys to move up, down, right, or left. The item moves a space at a time. To move an item in larger increments, hold the SHIFT key down while pressing the arrow keys.

Auto Positioning

Use this function to automatically reposition streams and unit operations. You select the object(s) for HYSYS to reposition. An internal algorithm determines the current location of objects on the PFD. This information is then used to set the most appropriate location for the selected item(s).

Auto-positioning will function differently depending on the object selected. For **unit operations**, HYSYS determines a new location for the

object and places it in that position. Manually moving streams attached to the unit operation after the auto-positioning has no effect on the position of the unit operation icon.

When a stream is auto-positioned, it becomes a floating icon **until** it is manually moved. This means that the stream is initially positioned by HYSYS after the Auto-Positioning function is accessed. If a unit operation icon is attached to the stream is then moved, the stream will be automatically repositioned, floating with the unit operation icon. As soon as you manually move a selected stream, the auto positioning function becomes inactive. Any subsequent movement of an operation to which this stream is attached will not affect the position of the stream. You will need to use the Auto Positioning function again to have the stream move with the operation.

You can access Auto-Positioning in the following ways:

Method	Description
Menu Bar	Under **PFD,** select either **Auto Position All** or **Auto Position Selected**
Object Inspection	Choose **Auto Position** from the Menu; only selected items are repositioned

Aligning Icons

This function can be used to horizontally or vertically align any combination of streams and objects on the PFD. Each object has a predetermined point through which it will be aligned.

To align icons, follow this procedure:

1. Select the objects that you want to align. At least two objects must be selected.
2. Object inspect the icon that is to be the anchoring point (all other selected objects will be aligned with this icon).
3. Choose the **Align Selected Icons** option from the object inspection menu.

Auto Snap Align

The Auto Snap feature on the PFD will automatically align objects if they are moved within an internally set tolerance. This feature helps eliminate the irregular line segments which might occur for streams.

The Auto Snap feature is always active and cannot be toggled on and off. This may pose a problem if you want to move an icon via the keyboard arrow keys. If HYSYS detects that the object is within the set tolerance, the object will initially move in the direction of the arrow key, but will snap back to its original position. To overcome this detail, use the SHIFT key with the keyboard arrow key.

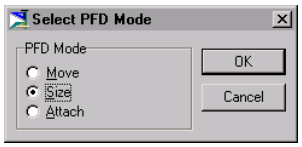
Sizing Objects

Unit Operations and streams installed on a PFD are a standard size. Sometimes the need arises to change the size of these objects, in which case you must be in Size Mode. The easiest way to do this is to press the ***Size Mode*** button in the upper left hand corner of the PFD.

Size Mode Button

You can also enter Size Mode from the **PFD** option in the Menu Bar. From this menu, choose **Select Mode**. The **Select PFD Mode** dialog appears, from which you can choose the **Size** radio button.

Figure 3.32

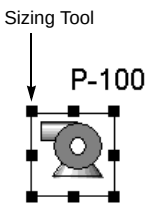


The third method to enter Size Mode is through Object Inspection of the PFD. Choose **Mode** and then **Size** from the menus that appear.

Once in Size Mode, the procedure for sizing is:

1. Select the object you wish to resize.
2. A white outline with eight target squares appears around the icon. These identify the directions in which you can size the icon.

Figure 3.33



3. To change the size of an icon, place the mouse pointer on top of the target squares. The pointer changes to the sizing tool (line with an arrowhead at each end) that indicates the directions that the object can be stretched or compressed.
4. Press the primary mouse button and drag away from the icon to enlarge, and towards the icon to shrink. The icon is sized according to the change in size of the box.

You can only size Labels and Annotations horizontally and to a maximum size. This is useful for text containing more than one line. You can size such a text box so that all text is shown on one line.

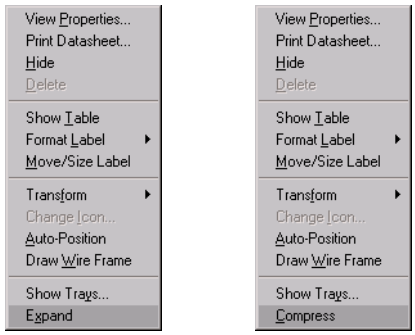
The Pump shown in Figure 3.33 is ready to size. However, the name associated with it, **P-100**, will not be resized. It remains at its default size although the size of its associated icon will change. You can change the Font size of the text to modify the size of the label. See the section on ***Moving and Sizing Labels.***

Rather than sizing several icons individually, you can size multiple icons simultaneously. Enter **Size** mode and select the icons you wish to size (use one of the multiple selection methods, explained previously). You will notice that each icon has its own outline around it. Pick any one of these boxes. Position the mouse pointer on one of the target boxes, and resize the icon. HYSYS sizes all the selected icons by this factor.

Column Tray Section Display

Once inside the Column PFD, you can Object Inspect the individual components of the Column, i.e., Tray Section, Condenser, and Reboiler. Object Inspection of the Tray Section provides two additional options, **Show Trays** and **Compress/Expand**.

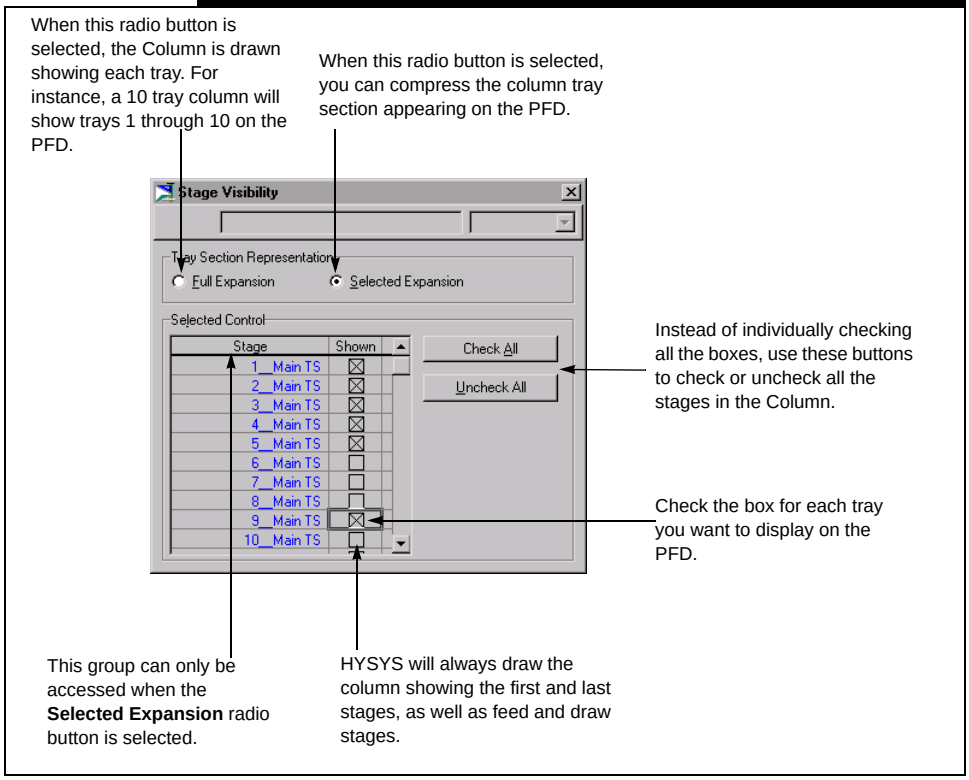
Figure 3.34



*If the tray section shown is compressed, the command at the bottom of the menu will be **Expand**. If the section is fully expanded, the command will be **Compress***

Option	Description
Show Trays	This option produces the **Stage Visibility** view, as shown here. This allows you to determine how the Column Tray Section will be displayed on the PFD.
Compress/Expand	This is a Toggle feature, so only one of these options appears in the object inspection menu at a time. For the following description, refer to the **Stage Visibility** dialog, shown in **Figure 3.35**. When the **Selected Expansion** radio button is chosen and the tray section is compressed, the **Expand** option appears in the menu. This allows you to expand the Column to full size (showing all the trays) without returning to the **Stage Visibility** dialog to select the **Full Expansion** radio button. After expanding the Column, you can return it to its selected compressed size by choosing **Compress** from the menu.

Figure 3.35



Swap Connections

Swap Connections Button

This function allows you to select two streams attached to the same object, and swap their nozzle connections. This is particularly useful when streams cross each other. There are a few ways in which you can access the **Swap Connections** function:

- Press the ***Swap Connections*** button.
- Object Inspect a connection point and select Swap Attachments from the menu.
- Select Swap Connections under PFD in the Menu Bar.
- Use the keyboard command **F**, which activates the Swap Connections button. The **ESC** key reverses the effect of the **F** key.

Once you have accessed the Swap Connections function, follow this procedure:

Available to Swap Connections

Unavailable

1. Notice the mouse pointer when it is over an area of the PFD. It takes on a special flip stream appearance, indicating which streams are available for the operation (arrow with a check mark and a numeral 1).
2. Move to the first stream you want to swap and select it with the primary mouse button. The cursor will change (replacing the 1 with a 2).
3. Select the second stream you want to swap.
4. After the swapping is complete, the ***Swap Connections*** button is released.

Note that HYSYS indicates an unacceptable choice for Swap Connections by replacing the check mark with an X.

3.10.2 Transforming Icons, Labels and Annotations

Using the Rotate and Mirror functions for multiple objects will change all selections in the same manner.

When you add a Unit Operation, Stream, or Annotation to the PFD, it always appears as it is shown in the Object Palette. You may want to alter the orientation of an icon to improve PFD clarity. This can be done through rotating and mirroring.

Method	Description
Transform >>Rotate	Rotate the icon of a selected object (clockwise) about its centre in one of three ways, 90 degrees, 180 degrees, and 270 degrees.
Transform>>Mirror	Mirrors the object about the X or Y axis.

Keyboard commands for selected objects:

X mirror about X axis

Y mirror about Y axis

***1** rotate by 90*

***2** rotate by 180*

***3** rotate by 270.*

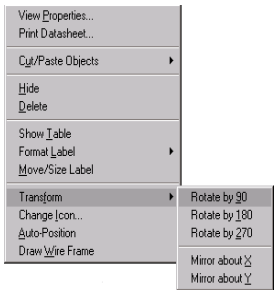
N returns original orientation

You can access the rotate and mirror functions through **Object Inspection**.

1. Select the icon(s) of choice.
2. Press the secondary button to produce the **Object Inspection** menu.

3. Select the **Transform** option. This produces an expandible menu containing the **Rotate** and **Mirror** functions. Your selection will be performed on the selected icon(s).

Figure 3.36



Changing Icons

HYSYS always chooses the default icon for PFD display. However, there are optional icons that you can use to represent the same Unit Operation. The method for changing icons is as follows:

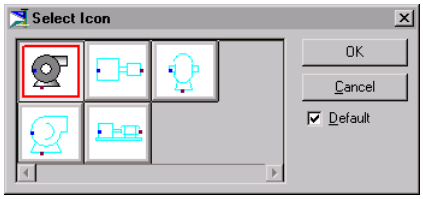
1. Select the PFD icon you wish to change.
2. Press the secondary mouse button to display the **Object Inspection** menu.
3. Choose the **Change Icon** option (if there are no alternate Icons, this option will be disabled in the menu).

Figure 3.37

4. Once you choose the **Icon** option, HYSYS displays a screen of all alternate icons allowed for this operation. Choose the one you want to use. Figure 3.38 shows the options available for the **COOLER** operation.

*Check the **Default** box if you want the selected Icon to be the default for your simulation.*

Figure 3.38



5. Notice that you also have the option of setting the new icon as the default. If you choose the default option, the new icon will be shown whenever you add this operation to the PFD.
6. Press the ***OK*** button to return to the PFD.

If you have more than one icon selected in the PFD, and they represent the same type of operation, you can change all selections to an alternate icon at the same time. If you have selected more than one type of operation, the **Change Icon** option will not be available in the Object Inspection menu.

Wire Frame / 3D Icons

When you Object Inspect a three dimensional icon, the **Draw Wire Frame** option is available in the menu. Any operation or stream whose icon is three dimensional can be transformed to an outline representation, or a **Wire Frame** view. A Wire Frame view of the Cooler is shown in Figure 3.39. Since this is a toggle function, the object inspection menu of a wire frame object will contain the option **Draw 3D,** which draws the icon in its default 3D view.

Figure 3.39

Cooler 3D Icon Cooler Wire Frame Icon

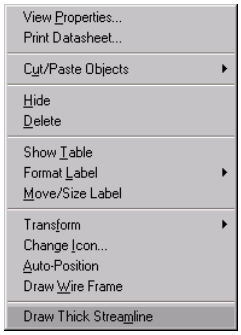
To switch all objects into wire frames, you can:

- Use one of the selection techniques described earlier to select all objects. Object inspect one object and choose the Draw Wire Frame option from the menu.
- Change the colour scheme to the HYSIM Colour Scheme (*refer to* **Section 3.9.1 - Selecting/Changing a Colour Scheme**, for details). The colour of individual items can be changed using this colour scheme.

Thick Stream Line

When object inspecting a stream icon, the **Draw Thick Streamline** option is available. This option creates a more visible stream by making it wider than the other streams. Use this option to simplify the task of tracking one particular stream in a complex Flowsheet (i.e. a pipe network). Return a stream to its default thickness by object inspecting the stream and selecting **Draw Normal Streamline** from the menu.

Figure 3.40



3.10.3 Auto-Scrolling

Unlike most of the features available on the PFD, the auto-scrolling function cannot be accessed through the main menu or by the keyboard. Only the mouse can be used.

Auto-scrolling enables continuous horizontal or vertical scrolling of the active pane, depending on the location of the mouse. To initiate auto-scrolling, place the mouse pointer anywhere in the active view and hold down the primary mouse button. Move the cursor near the PFD pane boundary. When the cursor enters a 15 pixel boundary at a pane edge and remains anywhere within the boundary for 0.3 seconds, the view of the PFD will begin to scroll. The closest boundary to which the cursor is positioned determines the direction of scrolling, as evidenced by the

movement of the scroll button (i.e. if the cursor is dragged to the top boundary, the view will scroll from its previous position and proceed upward while the scroll button moves up)

The speed of scrolling can be varied by moving the mouse within the boundary near the edge of the view. Faster scrolling will occur as the cursor is moved closer to the outside.

Mouse Wheel Scrolling

The Static PFD Scroll cursor appears when the mouse button is clicked.

The Scrolling Cursor points in the direction that the focus is moving.

As an alternative to using the scroll bars, you can scroll to any location on your PFD using the mouse wheel (or middle mouse button) functionality. Simply click on the mouse wheel while on the PFD. The cursor will change to the static PFD scroll cursor. Point your mouse in the direction you want to scroll. Once you have reached the location you wish to view, either click the mouse wheel a second time or place the cursor directly over the static PFD scroll cursor (which remains on your PFD until you click the mouse wheel).

3.10.4 Rerouting

Quick Route Mode

*The **Quick Route** function can be accessed while using any mode (i.e. Move/Size/Attach).*

To retain the clarity of the PFD, streams should not overlap unit operation icons. When working with large complex Flowsheets, each movement of an object causes HYSYS to reposition streams so that no unit operation icons are covered. If the PFD is complex, this repositioning can consume valuable computational time.

In **Quick Route** mode, the relocation and connection of objects is completed without considering the other objects in the Flowsheet. For example, if you move a valve, the valve icon and its associated streams are relocated without the repositioning of its streams, should one pass over another icon. After exiting **Quick Route** mode, HYSYS attempts to reposition streams so that icons do not overlap. The incurred benefit is that HYSYS repositions streams once rather than repeatedly doing so after each object is relocated.

Quick Route Button

To access **Quick Route** mode, press the ***Quick Route*** button from the button bar. After you have manipulated the objects on the PFD, press the ***Quick Route*** button again and HYSYS will ensure that no streams overlap unit operation icons.

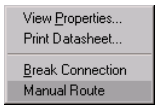
Manual Routing of Streams

There are two types of manual stream routing:

- **Moving the location of a horizontal or vertical line segment**
- **Adding bend points to create a new route for some portion of the stream**

Full Manual Route mode can only be accessed via the object inspection of a Stream, away from its icon. From the menu, you must select **Manual Route**. Only the portion of the stream which is object inspected becomes available for manual routing, i.e. you object inspect either the portion upstream or downstream of the stream icon.

Figure 3.41



*To interrupt **Manual Route** mode, simply click on an empty area of the PFD.*

In **Full Manual Route** mode, a bend point is shown at the end of each line segment and the portion of the stream available for manual routing changes colour. You can then manipulate any of the line segments in the selected Stream until Manual Route is interrupted.

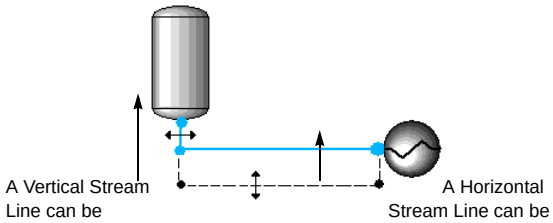
When not in **Manual Route** mode, you can still click on a stream line segment and drag that portion of the line. Notice that bend points are not shown in this case.

The procedures for the various Manual Routing processes follow.

Moving Line Segments:

1. To move a portion of a stream, object inspect it and select **Manual Route** from the menu. Anchor points will be shown at each corner of the stream route.
2. When you move the pointer over the stream, the pointer changes to a double arrow head cursor. This cursor is oriented in the direction that the portion of the stream can be moved, **Vertical** for a horizontal section and **Horizontal** for a vertical section.

Figure 3.42



3. With the primary mouse button, select the desired portion of the stream and drag it to the new location. As you move, a thin red line will be drawn indicating the new stream route.
4. When you release the mouse button, the stream is redrawn through the new path.

Adding Bend Points

If you wish to create a new route for a stream, you can add new bend points. The initial procedure is the same. Object inspect the stream and select **Manual Route** from the menu. HYSYS will redraw the stream with the existing bend points shown.

The logic behind the routing procedure is that you alternate between horizontal and vertical sections of line with each successive bend point.

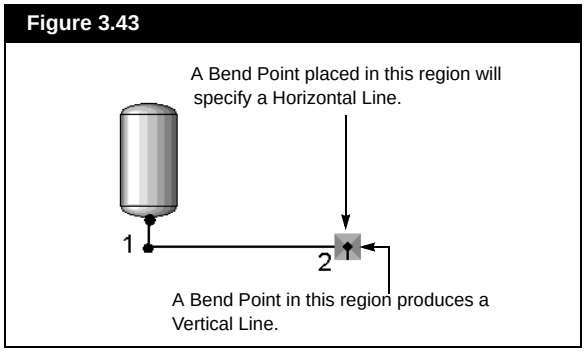
A bend point is an anchor that the stream passes through.

Use the ESC key to end Manual Routing. Use the DELETE key to delete a manual route in progress.

The idea of bend points is demonstrated in the Figure 3.43. Bend point 2 is the one we are going to manipulate. You have a choice of initially creating a horizontal or vertical line segment. Subsequent line orientation is determined by the first line, i.e. horizontal segments follow vertical and vice versa.

You can place a bend point by pressing the primary mouse button.

Figure 3.43



The mouse cursor will display a check mark at an acceptable location for the final Bend Point.

1. Select an existing bend point to begin the new route. The cursor shows as a '+' symbol when placed over a bend point, but is replaced by an '**X**' when the bend point has been selected.
2. Around a bend point are four regions, two of which define the next line as Horizontal, and two that define the next line as Vertical. As you drag the mouse pointer around the region of the bend point, notice that a light coloured line is displayed. It shows the area where the new line routing will be placed.

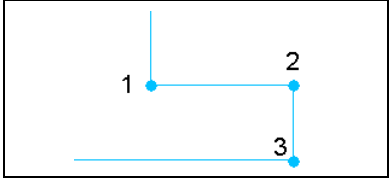
For slight kinks in a stream it may be easier to select the Stream icon, and use the keyboard arrow keys to nudge it into place, rather than inserting and removing bend points.

3. Click the primary mouse button to place a new bend point.
4. Continue to move the mouse pointer to the location of the next Bend Point and place it by pressing the primary mouse button.
5. The Manual Routing procedure can only be completed by placing the **Final Bend Point** on an **Existing Bend Point**. If this is not done, then the new routing you just laid out will not appear in the PFD. Any bend points added will also be erased.

Removing Bend Points

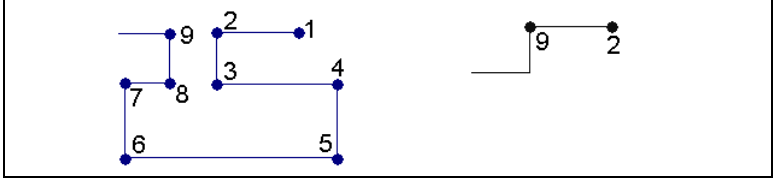
In certain instances, you may wish to remove bend points to provide a more direct route between operations. When there is a jog in the route, as shown in Figure 3.44, you can simply close down the section. Grab either the Horizontal section between points 1 and 2 and move it vertically until points 1 and 3 coincide, or grab the Vertical section (2 to 3) and move it horizontally until points 1 and 2 coincide. Either route will result in the jog being removed from the stream.

Figure 3.44



You can also remove several intermediate bend points. Select the bend point at the start of the section to be manipulated (in the case of Figure 3.45, point 2). Next, select the bend point at the end of the section (point 9) and double click with the primary mouse button on the end point. All intermediate bend points are removed.

Figure 3.45



Connection Line Straightening

This function removes all bend points from a stream to straighten the line between the stream icon and the unit operation.

To access the line straightening function, the orientation of the stream icon must be such that it can align with the nozzle connection, i.e. a horizontal stream icon is in alignment with a nozzle connected to the side of a unit operation.

To straighten a connection line:

1. Place the cursor over a stream and look for the cursor to change to a double-headed arrow
2. Object inspect the stream with the secondary mouse button.
3. Choose the **Straighten Stream Line** option from the pop-up menu. The stream section that will be straightened becomes a thin red line and the cursor changes to the acceptable/unacceptable connection indicator.
4. Move the cursor to either end of the red line where the cursor will change from an **X** to a check mark. This indicates an acceptable choice for line straightening.
5. Click the primary mouse button to straighten the line.

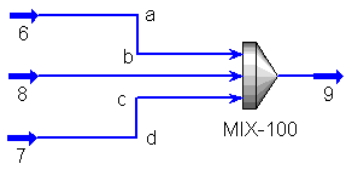
*The end of the thin red line which is **NOT** chosen serves as the anchor point, and will not move when the line is straightened.*

Line Segment Alignment

While performing manual routing, the mouse can be used to align sections of streams. Horizontal sections will be aligned horizontally and vertical sections will be vertically aligned. The streams need not be connected to the same unit operation, but the stream sections must be in close enough proximity so that the internal tolerance for the function is not exceeded.

As an example of line segment aligning, a unit operation with multiple feeds will be used. The Mixer, shown in Figure 3.46, has three feed streams, two of which contain vertical sections. For presentation purposes, the vertical sections of the streams are to be aligned. To align segments **ab** and **cd**, proceed as follows:

Figure 3.46



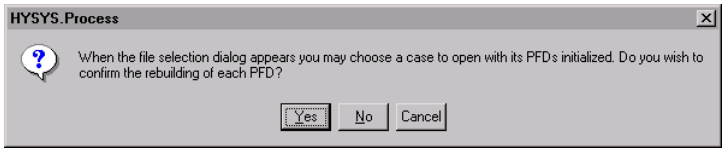
1. Object Inspect the line segment that will be moved. Choose **Manual Route** from the menu. The entire line will become highlighted and its bend points will be shown. For instance, object inspect line segment **ab** so that it will be moved to align with segment **cd**.
2. Use the primary mouse button to click on the anchoring segment, in this case segment **cd**.

The segments are now aligned. The same procedure is followed for the alignment of horizontal segments.

Rebuilding the PFD

In addition to all the different ways you can manipulate the PFD, HYSYS provides you with an option to rebuild the PFD for any simulation case. When you want to rebuild the PFD for a case, before opening that case, use the hot key combination CTRL SHIFT K Z. HYSYS will then display a message asking you if you want to rebuild the PFD.

Figure 3.47



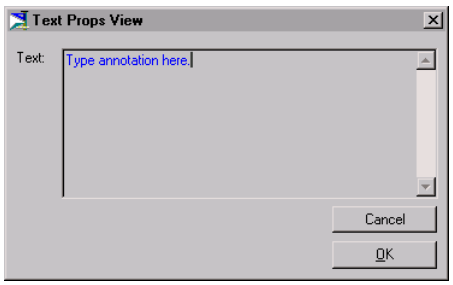
Click on the ***Yes*** button and then the **Open Simulation Case** view will appear. Proceed to open the case whose PFD you want to rebuild. If there are more than one PFD (in the main and sub-flowsheet environments), HYSYS will ask you if you want to rebuild each PFD (in the main and sub-flowsheet environments). Click on the ***Yes*** if you want to rebuild the specified PFD and ***No*** if you don't. If there is only one PFD (i.e. no sub-flowsheets) HYSYS will only ask you *once* if you want to rebuild the main PFD.

3.11 Adding Annotations

Add Text Annotation Button

Text can be placed on the PFD via the ***Add Text Annotation*** button. When this button is pressed, move the mouse pointer to the location on the PFD where you want to place the text and press the primary mouse button. HYSYS opens the **Text Props View**, in which you enter the text that will appear on the PFD. When you are done, press the ***OK*** button.

Figure 3.48



3.11.1 Editing Annotations

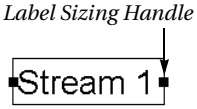
Object Inspect Text Annotations to access its change menu. The **View Properties** option displays the **Text Props View**. Other options include:

- **Hide**
- **Delete**
- **Transform — rotate by 90, 180 or 270 degrees**
- **Change Font**
- **Change Colour**

The listed functions for annotations are the same as those previously described for objects. The **Change Font** and **Change Colour** options are explained in the section ***Changing Text Fonts and Colours.***

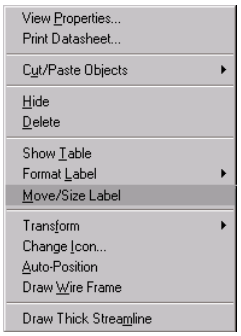
Moving and Sizing Labels

Label Sizing Handle



You can move and size object Labels on the PFD. Object Inspect an icon, and select **Move/Size Label** from the menu that appears. The object's Label is now available for moving and sizing. You can also free the Label by selecting the object and then pressing L on the keyboard. The same techniques mentioned for moving operations, streams and annotations can be used for labels.

Figure 3.49



Sizing Labels is handled a little differently. Labels can only be sized horizontally (notice the appearance of only two, not eight, resize handles on the box surrounding the label).

Once you select the **Move/Size Label** option, the Label is unlocked from the object to which it belongs. The Label relocks itself once it becomes deselected; however, you may select the Label's corresponding object

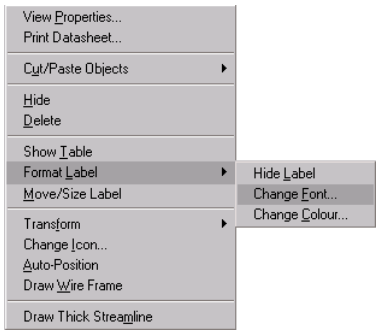
icon and then select the label again without relocking the Label. This facilitates moving and sizing both the icon and its Label at one time without repeatedly selecting the **Move/Size Label** command.

Changing Text Fonts and Colours

You can make global changes for Font and Colour through Session Preferences (Font and Colours pages).

When you Object Inspect a Label or Text Annotation, the menu provides options for changing the Font and Colour. When Object Inspecting a Label, you have to select the **Format Label** option to get at the **Change Font** and **Change Colour** features. When you Object Inspect an Annotation, directly select either the **Change Font** or **Change Colour** option.

Figure 3.50



To change the font or colour for multiple objects, select the entire group and then make the changes.

Menu Item	Description
Change Font	This option allows you to change the font for a Label or Text Annotation. This function is not global, so only the selected object's font is changed.
Change Colour	Accesses a Colour Palette, and allows you to change the colour of the selected Label or Text Annotation.

3.12 Hiding PFD Objects

Hiding a unit operation or stream on your PFD does not alter your simulation case.

A Show/Hide option also exists for displaying sub-flowsheet objects on the Main flowsheet PFD. For further details, see ***Section 3.8.2 - Accessing Column or Sub-Flowsheet PFDs****.*

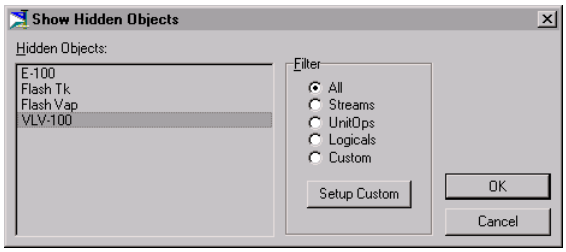
Any Object on the PFD can be hidden. You can select multiple icons to be hidden. Hiding an Icon does not prevent the case from solving. Hide operations, streams, and text annotations by selecting the **Hide** option from the Object Inspection menu.

To hide Labels, you need to select the **Format Label** option and then the **Hide Label** option in the Object Inspection menu.

Reveal Hidden Objects

Any objects that are hidden on the PFD can be revealed with this option. You can access this command in two ways. The first is through the **PFD** option in the Menu Bar, and the second using PFD inspection. When you choose **Reveal Hidden Objects**, the view in Figure 3.51 is displayed; it allows you to specify the hidden objects to be revealed.

Figure 3.51



3.13 Multi Pane PFDs

Since each pane is simply a different representation of the same Flowsheet, you can interact across views, i.e., connect an operation in one pane to an operation in another.

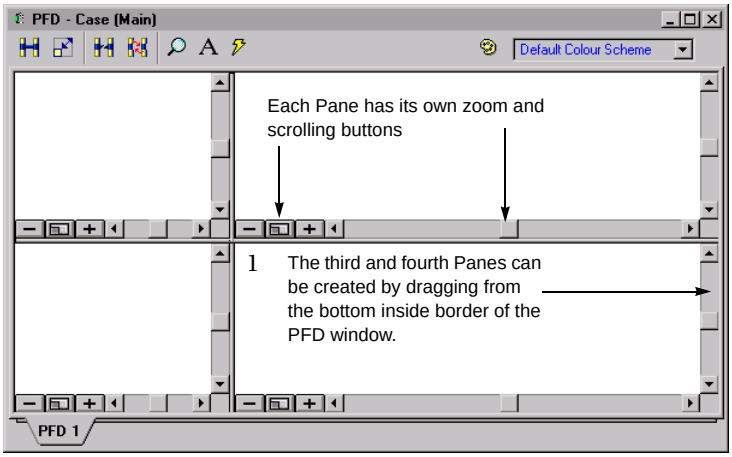
The PFD interface allows you to separate the PFD window into a maximum of four panes. Each pane contains the entire information regarding the PFD (operations, connections, etc.), but operates independently with regards to the area of focus, zooming etc.

New panes are created using the manipulation areas located on the border edge of the PFD view. Notice that the PFD appears to have two borders. The manipulation areas are located along the inner border. You can split the PFD once vertically and once horizontally, thus giving a maximum of four panes. Each new view that you create will have its own zoom and scroll buttons. This allows the various views to be at different zoom levels or locations in the overall PFD.

When the mouse pointer is over the heavy inside border of the PFD, the pointer changes into the multi-pane sizing tool. This is represented by the symbol shown to the side. The symbol is rotated 90 degrees for a horizontal split. The Status Bar in the bottom of the screen indicates which way the split will occur.

Multi-Pane Sizing Tool

Figure 3.52



There is no requirement on which way you initially split the PFD. For description purposes, a pane that is created by dragging vertically is termed a Horizontally split pane, and one created by dragging horizontally is termed a Vertically split pane.

Working Across Panes

By creating Split Panes within your PFD, you can focus in on different sections of the PFD in each pane.

The splitting of PFD panes may become useful in cases where the PFD is complex enough that you cannot view it in its entirety without making it very small. When working in split panes of a PFD, it is important to remember that all the panes interact with each other. This allows you to connect an operation or stream in one pane to an operation or stream in another. Whatever you do in one pane will be done in the overall PFD, so you can see the change in any of the panes.

Resizing or Closing Split Panes

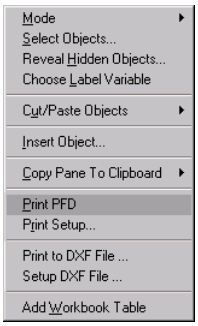
Once panes have been created, they can be resized or closed. To resize or close a Vertically split pane, place the cursor on the right side of the vertical scroll bar (in the split pane). The multi-pane tool will appear. Drag it left or right to change the existing view. Dragging to the extreme

right will close the split pane. The procedure is the same for a Horizontally split pane, except that you place the cursor on the bottom of the horizontal scroll bar and drag up or down. Dragging to the bottom of the original PFD window will close the split pane.

3.14 Printing the PFD

All items (Streams, Operations, Text, and PFD Tables) included within the PFD Window can be printed. HYSYS prints the PFD as it appears on the screen. The PFD can be printed from the PFD Inspection menu by selecting **Print PFD**.

Figure 3.53



When the PFD has focus, you can select **Print Snapshot** under **File** in the Menu Bar to print the PFD.

You can also access the **Print Setup** view from the **PFD Inspection** menu.

4 Workbook

4.1 Opening the Workbook 4

4.2 Installing a Stream or Operation 5

4.2.1 New Stream 5
4.2.2 New Operation 5

4.3 Accessing Streams or Operations 6

4.3.1 Viewing Sub-Flowsheet Objects 9
4.3.2 Show Name Only 10

4.4 Deleting Streams or Operations 10

4.5 Setup of the Workbook Page 11

4.5.1 Installing a New Tab in the Workbook 12
4.5.2 Editing a Workbook Tab 14
4.5.3 Deleting a Tab from the Workbook 18

4.6 Exporting/Importing Workbook Tabs 18

4.7 Sorting Information 20

The most concise method of displaying process information is in a tabular format. The HYSYS Workbook has been designed to satisfy this need, extending the concept to the entire simulation. In addition to displaying stream information and general unit operation information, the Workbook can also be configured to display information about any *object type* (streams, pipes, controllers, separators, etc.).

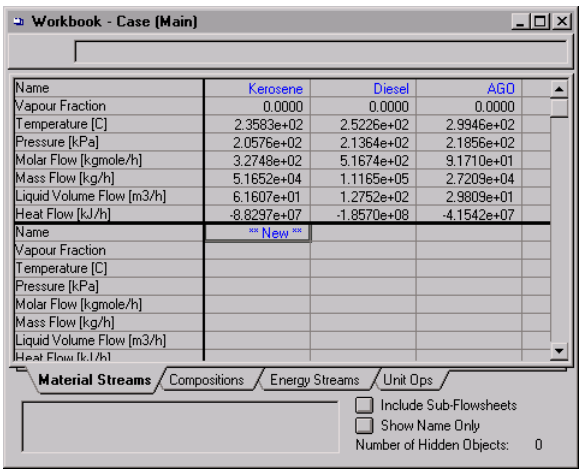
The Workbook becomes a collection of tabs. For example, suppose you add a tab for Separators. Every separator in the Flowsheet will be on that tab, with the current value of process variables appropriate to the separator being displayed for each. To provide the greatest degree of flexibility, you can modify the variable set to show whatever variables are of interest, or install multiple tabs for the same object type in varying levels of detail.

Not only is the Workbook extremely powerful for process analysis, but it has also been developed as an integral element in the building and manipulation of your simulation. In addition to displaying the process information, you can make changes to specifications directly from the Workbook and calculations will be performed automatically. Mechanisms have also been built into the Workbook to provide you immediate access to the property view for an individual stream or operation.

Use the CTRL W hot key to access a Workbook.

Each Flowsheet in your simulation (Main Flowsheet and Column/ Template SubFlowsheets) possesses its own Workbook. You can access the Workbook for any Flowsheet from any location in your simulation.

Figure 4.1

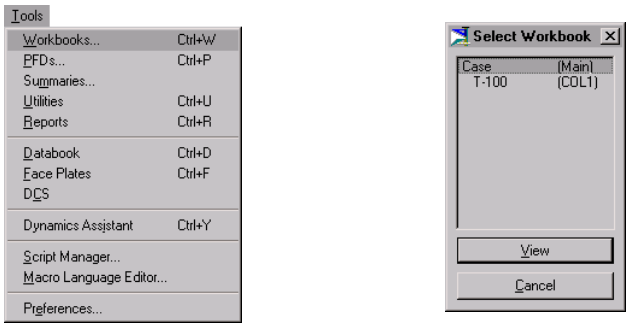


4.1 Opening the Workbook

Workbook Button

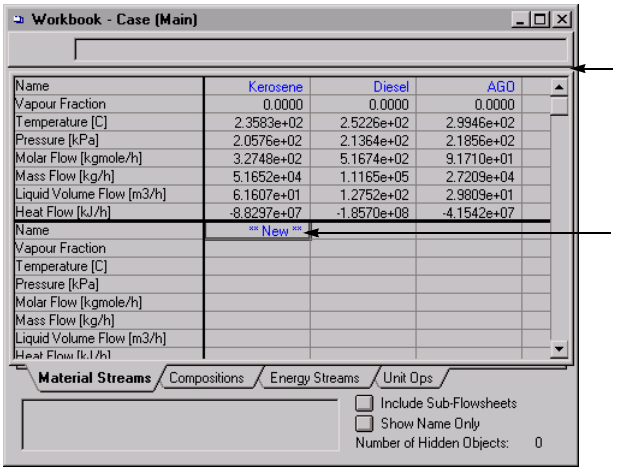
You can open the Workbook for the current Flowsheet by selecting the ***Workbook*** button. Alternately, you can access any Workbook in the simulation by selecting **Workbooks** under **Tools** in the Main Menu. The **Select Workbook** view displays all Flowsheets in the simulation. To open a specific Workbook, select its Flowsheet and press the ***View*** button or double click on the Flowsheet name.

Figure 4.2



The first time you access the Workbook, it will open to the **Material Streams** tab, displaying the basic stream information for all streams currently installed in the Main Flowsheet. Notice that the default Workbook also contains tabs for **Compositions**, **Energy Streams** and **Unit Ops**. Shown here is the default **Material Streams** tab.

Figure 4.3



4.2 Installing a Stream or Operation

4.2.1 New Stream

Refer to Section 1.3.4 - Installing Streams for more information.

The procedure for installing a new stream through the Workbook is as follows:

1. Select the **New** cell and begin typing in your stream name. HYSYS opens the Edit Bar at the top of the Workbook tab, echoing your input. Press ENTER to complete your input and transfer it into that cell.
2. HYSYS automatically advances to the next cell, Vapour Fraction. Continue supplying your input.
3. Supply input for any two of the three variables: Vapour Fraction, Temperature, Pressure, as well as a flowrate.
4. You can supply composition in numerous ways. You can provide input directly on the **Compositions** tab, access the property view for the stream, or double click on the appropriate flow cell to access the **Input Composition** dialog directly.

4.2.2 New Operation

Refer to Section 1.3.5 - Installing Operations for more information

If your Workbook tab lists operations of a specific type, then the same procedure as outlined above for streams applies for adding a new operation.

To install a Unit Operation from the Workbook, proceed as follows:

*If the unit operation category is known, the list can be filtered by selecting the appropriate radio button. For example, selection of the **Heat Transfer Equipment** radio button for a heater or cooler operation.*

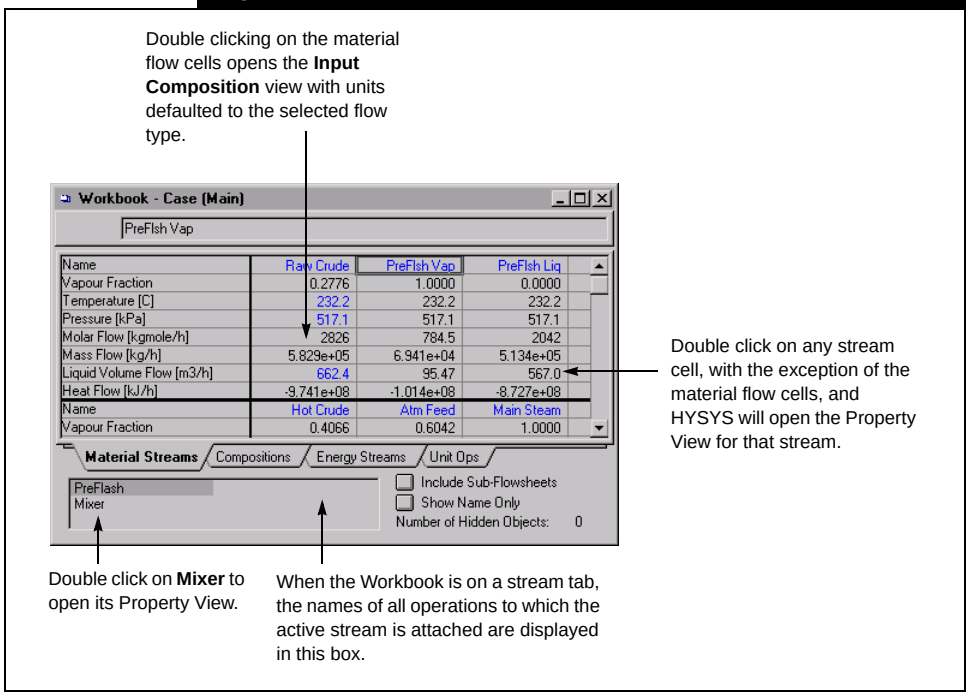
1. Select the **Unit Ops** tab to open the view.
2. To access the list of unit operations, press the ***Add UnitOp*** button or use the hot key **F12**.
3. Choose the unit operation from the **Unit Ops** dialog and press the ***Add*** button.

4.3 Accessing Streams or Operations

The functionality of HYSYS allows you to access both streams and unit operations from any of the default Workbook tabs.

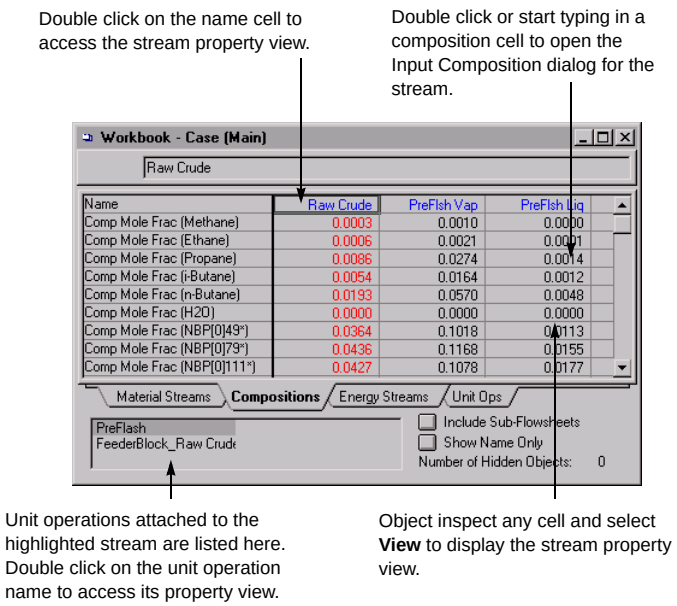
You can access the Property View for a material stream directly from the **Material Streams** tab. From this location, you can also access the **Input Composition** view, as well as open the property view for any operation attached to a stream.

Figure 4.4



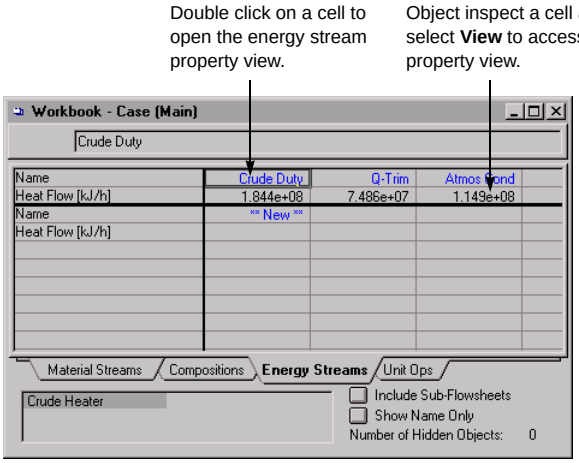
From the **Compositions** tab, you can access a stream property view, the Input Composition dialog for a stream or the property view of an attached unit operation.

Figure 4.5



You can access energy stream property views from the **Energy Streams** tab. The property views for unit operations to which the energy streams are attached can also be displayed via this tab.

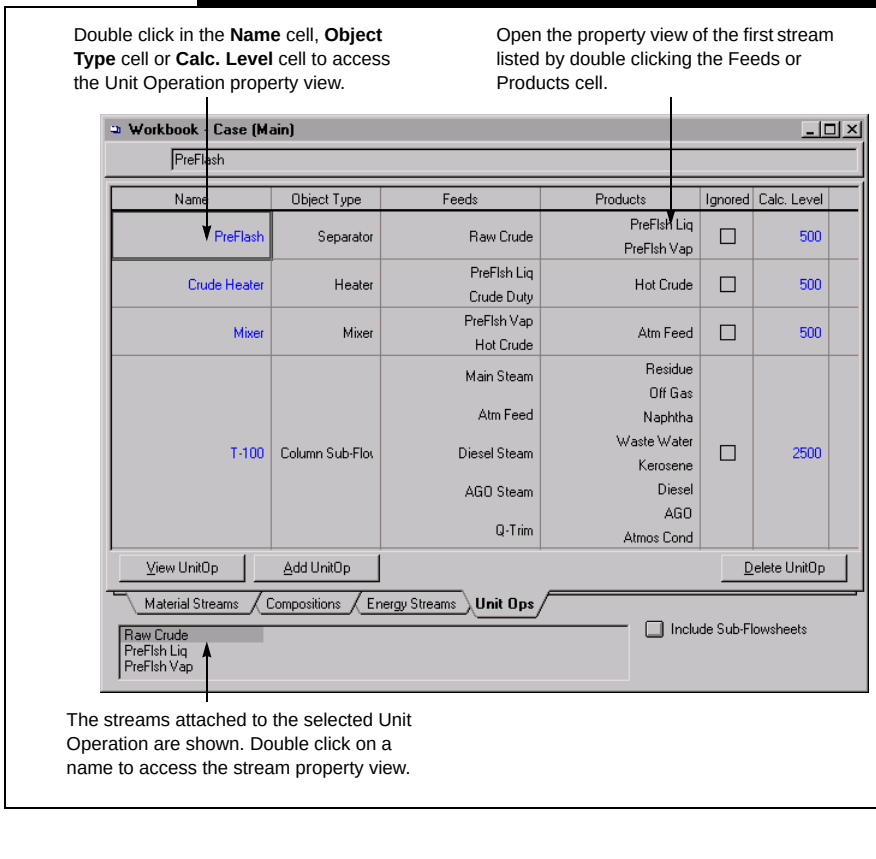
Figure 4.6



If you are on a Workbook tab of operations, the same capabilities apply. You can access each operation's property view by double clicking in an associated cell.

On the **Unit Ops** tab, the property view of any unit operation in the Flowsheet can be accessed. The property view of any stream which is attached to a unit operation can also be opened from this tab.

Figure 4.7



4.3.1 Viewing Sub-Flowsheet Objects

From the Workbook in the Main Environment, you can view information concerning Sub-Flowsheet items. You can display SubFlowsheet information by pressing the ***Include Sub-Flowsheets*** button, located in the lower right corner of the Workbook. For instance, on the **Material Streams** tab, you can view the Column Sub-Flowsheet streams such as **Reflux** and **To Condenser**.

Figure 4.8

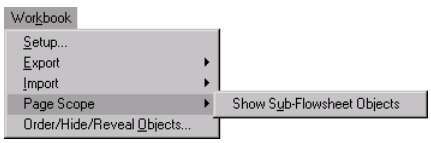
The command to display Sub-Flowsheet objects must be repeated on each individual Workbook tab.

The functionality of the ***Include Sub-Flowsheets*** button is not global to the entire Workbook. With each tab, you have the option of including the Sub-Flowsheet items.

To hide the Sub-Flowsheet items from a Workbook tab, simply press the ***Include Sub-Flowsheets*** button again.

You can also view or hide Sub-Flowsheet items on a Workbook tab by using the Main Menu. Under **Workbook**, select **Page Scope** and from the submenu, choose **Show/Hide Sub-Flowsheet Objects**.

Figure 4.9



Note that when the ***Include Sub-Flowsheets*** option is activated, the **Composition** tab will only display results if a common Fluid Package is shared by the Sub-Flowsheets and the Main Environment in the case. Since different Fluid Packages can contain both different types and numbers of components, it is not possible to display the compositions in the same form.

4.3.2 Show Name Only

*The **Show Name Only** button is not available on the default **Unit Ops** page.*

To simplify the search for a particular stream or unit operation, select the ***Show Name Only*** button, located in the lower right corner of the Workbook. This button hides all object data except names. You can then place the cursor on the name of the object of choice and press the ***Show Name Only*** button again. All object data reappears and the cursor remains on the selected object.

Figure 4.10

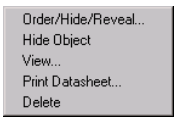
The functionality of the ***Show Name Only*** button is not global to the entire Workbook. With each Workbook tab, you have the option of showing either the names or all object data.

4.4 Deleting Streams or Operations

*To delete objects without confirmation, clear the **Confirm Delete** check box on the **Simulation** page of the **Session Preferences** view. Access **Preferences** under **Tools** in the main menu.*

You can delete streams directly from the Workbook by object inspecting on any cell associated with the stream. A pop-up menu will appear with the option to view or delete the stream. Choosing **Delete** will display a confirmation box to ensure that the choice was not accidental.

Figure 4.11



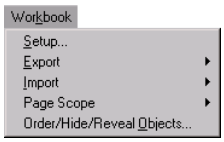
An alternative to using object inspection is to highlight the name of the stream and press the DELETE key on the keyboard.

For unit operations, you can use either of the available deletion methods unless the default **Unit Ops** page is being displayed. In this case, highlight the operation and select the ***Delete*** button.

4.5 Setup of the Workbook Page

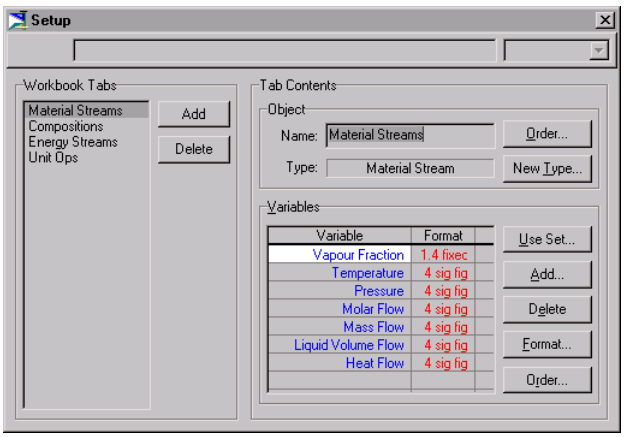
When the Workbook is active, HYSYS will redraw the Menu Bar at the top of the Desktop, adding the **Workbook** item. Select **Workbook** or use the hot key combination, ALT K to open the Workbook drop down menu.

Figure 4.12



Selecting the **Setup** option displays the following dialog box:

Figure 4.13



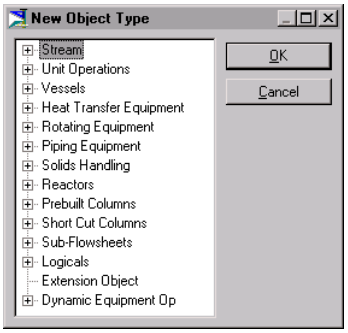
4.5.1 Installing a New Tab in the Workbook

*You can also access the **Setup** page by object inspecting a Workbook tab and selecting **Setup** from the menu.*

When a new tab is added to the Workbook, it is inserted directly before the highlighted tab in the **Workbook Tabs** group of the **Setup** view.The procedure for adding a New Tab to the Workbook by installing a customized Stream tab is illustrated:

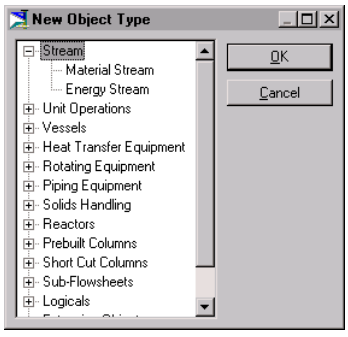
1. Select **Workbook** from the Menu Bar, and then select **Setup**. This opens the **Setup** dialog.
2. Highlight the default **Material Streams** tab in the **Workbook Tabs** group.
3. Select the ***Add*** button in the **Workbook Tabs** group. The **New Object Type** view is automatically opened. The view that opens shows 13 options. The '+' symbol indicates that there are additional sub-items underneath. To open the list of sub-items, double click on the object, or click on the '+'. Once the list is open, a '+' symbol will be replaced with '-'. To close the list, double click on the main option, or click on the '-' symbol.

Figure 4.14



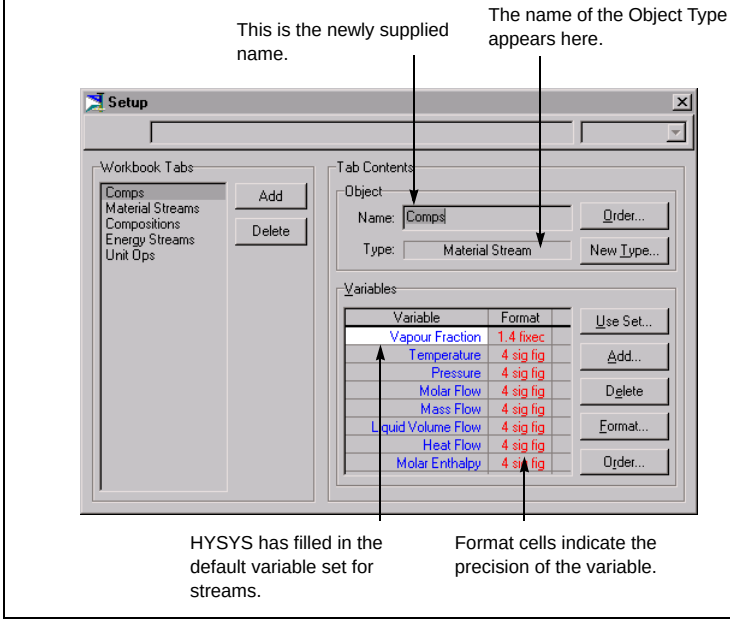
4. Double click on the **Streams** item. The list is redrawn as shown. Notice that Material and Energy Stream options have been added to the list.

Figure 4.15



5. Select **Material Stream** as the Operation Type and then press the ***OK*** button. You are returned to the **Setup** view, and the Material Stream default variable set has been added as a Workbook tab.
6. Notice in the **Tab Contents** section that a name has been chosen, in this case **Material Streams 2**, based on the selected object type. Move the cursor to the **Name** cell and enter in a more descriptive name for the tab, **Comps**.

Figure 4.16



Pressing the ***Close*** button will return you to the Workbook, with it open to the tab you just created. The following section continues with the variable set customization.

4.5.2 Editing a Workbook Tab

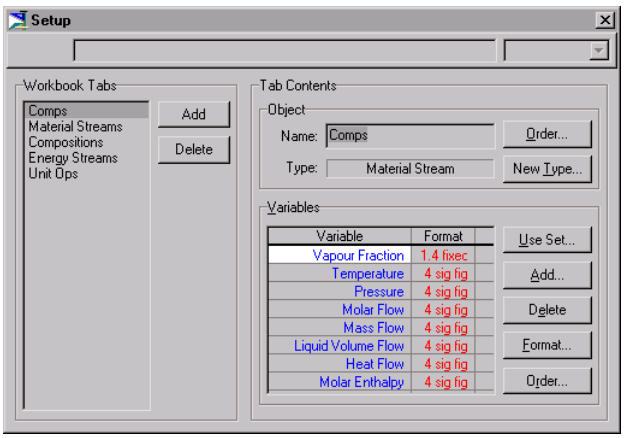
*You cannot make changes to the default **Unit Ops** Workbook page.*

From the **Setup** dialog, not only can you add new tabs to the Workbook, but you can also edit existing tabs. The **Setup** dialog is used to add, delete, and sort variables on the Workbook tab. Use can be made of default variable sets or you can individually select the variables to display. You can also designate the format of each variable here. The name and type of tab can be modified, as well as the order of objects on the tab.

The following procedure, which will focus on the new Workbook tab created in the previous section, will demonstrate some of the available editing features:

1. Open the Workbook tab you want to edit by selecting the appropriate tab in the Workbook. In this case, select the **Comps** tab.
2. Select **Workbook** in the Menu Bar, and then choose **Setup**. The **Setup** dialog appears. Note that the highlight in the **Workbook Tabs** group is on the active Workbook tab. The **Tab Contents** group contains the information concerning the active tab, including its name and type and all variables listed on the tab.

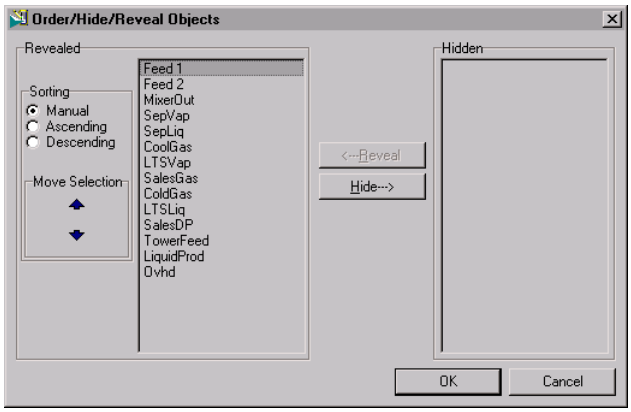
Figure 4.17



3. In the **Object** group, you can press the ***New Type*** button to change the type of Workbook tab. The **New Object Type** dialog appears, from which you can select a new tab type. For this example however, the **Material Stream** option is satisfactory.
4. Also in the **Object** group of the **Setup** dialog, you can select the ***Order*** button. From the **Order/Hide/Reveal Objects** dialog, you can arrange the order of the objects as they will be shown on the Workbook tab. Multiple selections can be made in the **Revealed** list.

*Refer to Section 4.7 - Sorting Information for details on the **Reorder Objects** dialog.*

Figure 4.18



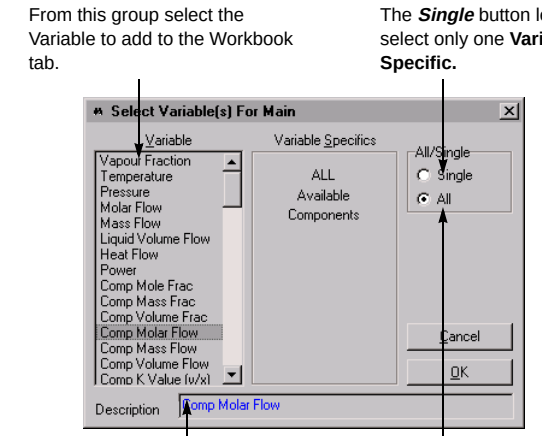
5. After you have finished rearranging objects, press the ***Close*** button.
6. Focus will now be shifted to the variables contained on the Workbook tab. To remove a variable, highlight it in the **Variables** group and press the ***Delete*** button.

You can remove multiple variables from a Workbook tab. Refer to Section 1.1.9 - Selecting Items.

7. To add variables to the tab, select the ***Add*** button. This produces the **Select Variables** dialog, which contains appropriate variables for the Object Type.

*Press the **Use Set** button for default variable lists.*

Figure 4.19



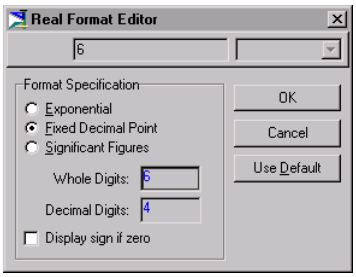
You can only add one Variable at a time.

8. From the **Variable** column, select the variable to add to the tab. In the above view, the Variable **Comp Molar Flow** is selected.
9. If the selected Variable has specifics, such as components, select these from the **Variable Specifics** column.
10. Select the ***OK*** button to return to the **Setup** view.
11. You can specify the display precision of a variable by highlighting it and pressing the ***Format*** button. You can specify the number of significant digits, a fixed number of decimal places or have the

*Refer to **Section 6.5 - Format Editor** for details on the Real Format Editor.*

variable appear in exponential form. Make changes and press ***OK*** or choose the ***Use Default*** button for the choice recommended by HYSYS.

Figure 4.20



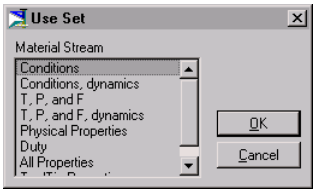
See Section 4.7 - Sorting Information for a more detailed explanation.

12. Having returned to the **Setup** dialog, choose the ***Order*** button if you wish to change the order in which the variables appear on the Workbook tab.
13. Select the ***Close*** button on the **Setup** dialog to return to the Workbook tab.

Using Default Variable Sets

If you do not want to create your own variable set by using the ***Add*** and ***Delete*** buttons, you can choose one of the predefined variable sets for your Workbook tab. Selecting the ***Use Set*** button on the **Setup** dialog accesses the view shown here. The list of default sets will depend on the page Object Type.

Figure 4.21



These sets may serve as the basis for user modified variable sets.

4.5.3 Deleting a Tab from the Workbook

There is no confirmation message when you delete a Workbook tab. You cannot recover deleted tabs.

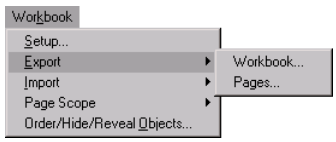
To **Delete** a tab from the Workbook, select the **Workbook** option from the Menu Bar and then select **Setup**. Once the Setup view is shown, choose the tab from the **Workbook Tabs** group and press the ***Delete*** button. The Workbook tab is now deleted.

4.6 Exporting/Importing Workbook Tabs

Once a Workbook has been customized with the addition/deletion/sorting of tabs and variables, it can be exported to a file. This same workbook setup can then be used in another simulation case by importing this file into HYSYS.

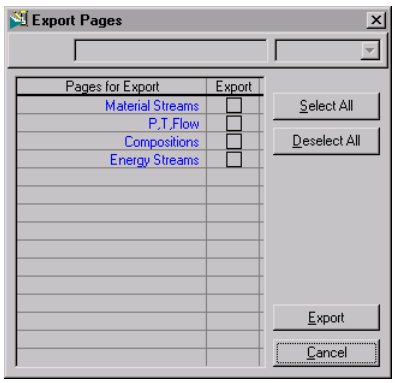
Access the exporting feature by selecting **Export** under **Workbook** in the Menu Bar. From here you can choose to either export the entire workbook or just some of the pages in the workbook.

Figure 4.22



If you choose to export pages the following dialog will appear allowing you to specify what pages to export.

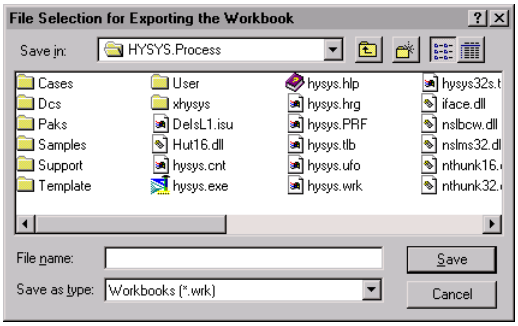
Figure 4.23



After you have made a selection, either workbook or pages, choose a **File Path** (the default is usually satisfactory), a **File Name** and select the ***OK*** button.

The format for the .wrk files does not change for an exported workbook or individual exported pages. This means that existing .wrk files are completely backwards and foreards compatable, as well as interchangeable for workbook or workbook page importing

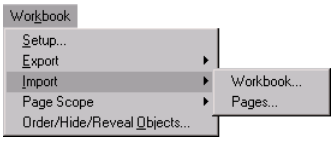
Figure 4.24



Access the importing feature by selecting **Import** under **Workbook** in the Menu Bar. From here you can choose to either import the entire workbook or just some of the pages in the workbook.

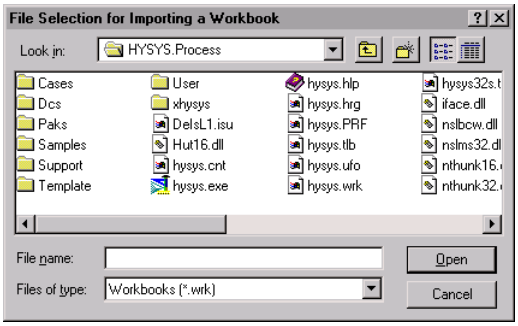
A unique-naming function has been added to distinguish imported pages from pre-existing pages that have the same name.

Figure 4.25



Choose the Workbook file from the list and select the ***OK*** button. If the file is not listed in the **File Name** box, an alternate **File Path** may be needed.

Figure 4.26



4.7 Sorting Information

You can change the views for opening and saving files to the Windows standard in the Preferences view.

Each Workbook tab can be sorted independently. You can sort the objects on a tab or you can select the order of the variables on a tab.

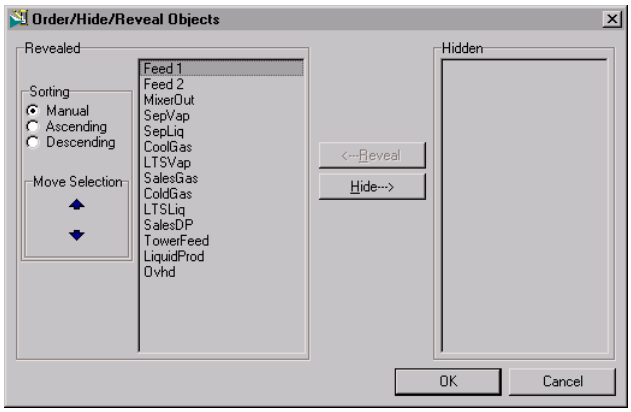
Object Sort

There are three locations where you can access the Sort function, all three display the **Order/Hide/Reveal Objects** view:

- **Order/Hide/Reveal Objects** in the **Workbook** Menu
- **Order/Hide/Reveal Objects** in the Object Inspection of the Workbook tabs or any cell
- ***Order*** button in the **Object** group of the **Workbook Setup** view.

If the Workbook tab is sorted alphabetically, objects continue to be sorted automatically following any ensuing Name changes.

Figure 4.27



Option	Definition
Manual	Allows you to manuall rearrange the Workbook objects. Select an object (or multiple objects) in the **Revealed** group and use the up and down arrows to move the selected object(s) through the list.
Alphabetical Ascending	Sorts the Names of the objects in alphabetically ascending order. Any item with a numerical name will be listed first in an Alphabetical Ascending Sort.
Alphabetical Descending	Sorts the names of the objects in alphabetically descending order. Lists any items with a numerical name last.

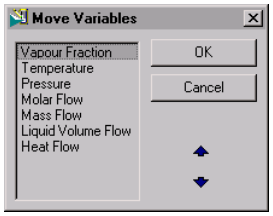
Use the ***Hide*** and ***Reveal*** buttons to hide or display the objects within the workbook.

Variable Sort

The Variable Sort lets you reorder the variables on the Workbook tab. For this example, you will use the default stream variables list. To reorganize the order in which these variables appear on the Workbook tab, use the following procedure.

1. Select **Setup** from the **Workbook** drop down menu.
2. On the **Setup** view, select a Workbook tab from the **Workbook Tabs** group and press the ***Order*** button in the **Variables** group. The **Move Components** dialog appears.

Figure 4.28



If variables were added to the Workbook tab as a group, i.e. component molar flows, you cannot move these individually, but only as a group.

3. Select the variable(s) you want moved from the **Move Variable** group.
4. Press the ***Up Arrow*** or ***Down Arrow*** button, depending on how you want to move the variable. The variable(s) selected in the **Move Variable** group will be in a new position. The other variables will not have changed.
5. Close the view by pressing the ***Close*** button and then press the ***Close*** button in the **Setup** view to return to the Workbook tab.

5 Flowsheet Analysis in HYSYS

5.1 Property View Flowsheet Analysis 3

5.1.1 Stream Analysis 4
5.1.2 Unit Operation Analysis 5
5.1.3 Simulation Navigator 7

5.2 Navigation 8

5.2.1 Object Navigator 9
5.2.2 Variable Navigator 11

5.3 DataBook 15

5.3.1 Variables Tab 16
5.3.2 Process Data Tables Tab 17
5.3.3 Strip Charts Tab 20
5.3.4 Data Recorder Tab 33
5.3.5 Case Studies Tab 37

5.4 Object Status Window/Trace Window 46

5.4.1 Opening and Sizing the Windows 46
5.4.2 Available Information 46
5.4.3 Object Inspection Options 47

5.5 Attaching Utilities 48

5.5.1 Adding a Utility from a Stream Property View 49
5.5.2 Adding a Utility From the Flowsheet 50
5.5.3 Viewing Utilities 52
5.5.4 Deleting Utilities 53
5.5.5 Attaching a Utility to a Different Stream 53

HYSYS has been designed such that the analysis of your process is not something that occurs after the fact, but rather, it is an integral part of building the model. This is a direct result of the power of **Event Driven** simulation. Information is processed as you supply it, with calculations being performed automatically. The key is that results are available immediately, and due to the Event Driven nature of HYSYS, you can access whatever information you need at any time, including as calculations are proceeding.

The approach incorporated for Flowsheet analysis in HYSYS is very powerful. You gain the greatest possible process understanding by examining both the individual components of your process and the process as a whole. All internal relationships which govern your process are revealed, rather than just the end results.

The Flowsheet analysis tools in HYSYS have been developed around this interactive approach to modelling. Because of the object oriented design, there is a great degree of flexibility provided in this area. Since there is no standardized way to look at process information, there are a variety of ways in which the necessary information can be presented, from tabular to plots. The power of HYSYS is demonstrated in that the various analytical tools are available simultaneously, with the same information being displayed in a variety of manners.

Previous chapters have already outlined some of the analytical capabilities built into the Main Interface elements, such as the Property View, the PFD and the Workbook. This chapter will explain some of these in greater detail as well as the specific analytical tools available.

5.1 Property View Flowsheet Analysis

In HYSYS, Stream and Operation property views contain analytical information based on the current Flowsheet conditions. For example, the Stream property view has a page which contains information concerning all phases present in the stream. Also, certain operations have pages which display performance profiles, results, and other analytical information.

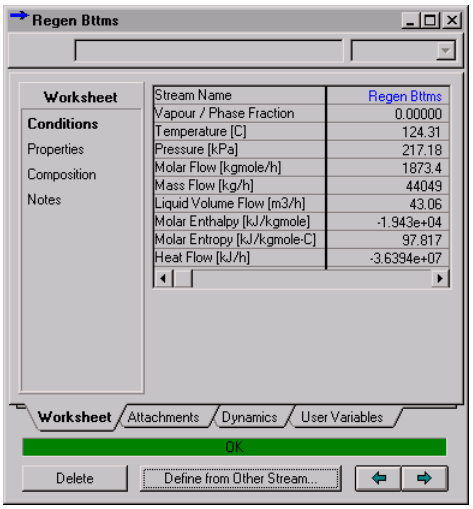
5.1.1 Stream Analysis

*Refer to Section 2.1 - Material Stream Property View in the **Steady State Modeling** manual for details on the various page tabs.*

Included as part of the Stream property view are two tabs containing information pertinent to stream analysis: the **Worksheet** and **Attachments** tabs. The **Properties** page in the **Worksheet** tab contains detailed physical property information about the stream. The **Conditions** page is simply a subset of the information provided in the **Properties** page. The **Utilities** page in the **Attachments** tab is used to attach Utilities to the stream, while the **Unit Ops** page indicates the unit operations attached to the stream.

See Section 5.5 - Attaching Utilities - for information on attaching a utility to a stream.

Figure 5.1

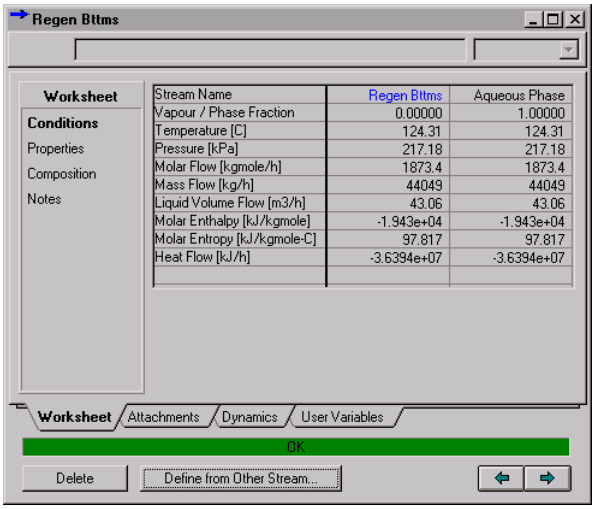


Open the Stream property view and go to the **Properties** page in the **Worksheet** tab. In addition to containing the basic stream conditions, more detailed physical property information for the stream is shown.

With the Stream property view at its default size, the page has vertical and horizontal scroll bars. The vertical scroll bar allows you to view the additional properties in the list. By using the horizontal scroll bar, you can scroll left and right to view the Vapour, Liquid and/or Aqueous phases for the stream.

Instead of scrolling through the view, you can resize it so that all phases, and all of the properties for each phase can be seen, as shown below. The Liquid Phase is referred to as the Aqueous Phase because Water is present in the Stream. The other phases you may encounter are Light and/or Heavy Liquid.

Figure 5.2



5.1.2 Unit Operation Analysis

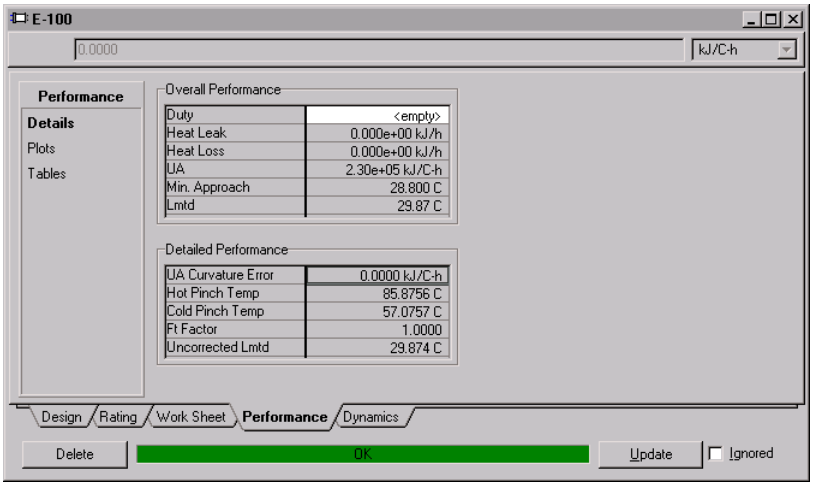
*A **Worksheet** tab is available on each unit operation property view. It provides access to the streams attached to the unit.*

Many Unit Operations in HYSYS have pages that contain analytical information. The Heat Exchanger displays its analytical information on the **Worksheet** and **Performance** tabs.

The **Details** page in the **Performance** tab is shown in Figure 5.3. This page displays Heat Exchanger information in two groups:

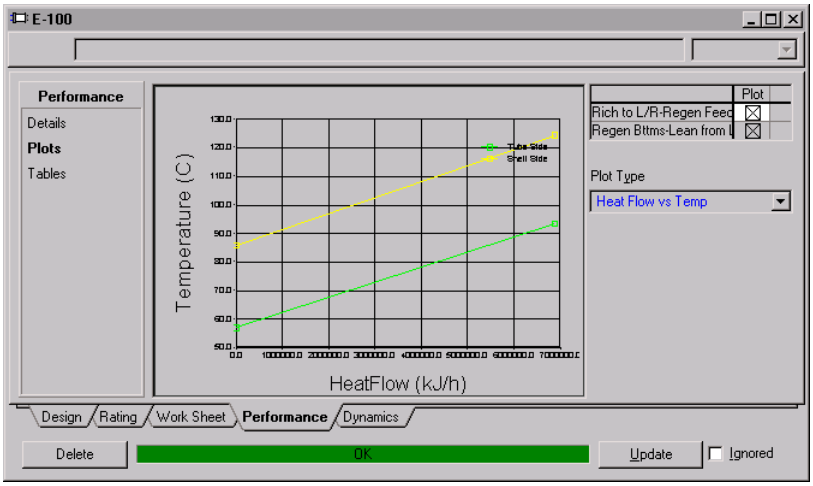
- **Overall Performance**
- **Detailed Performance**

Figure 5.3



The **Plots** page in the **Performance** tab lets you to generate curves for the shell and/or tube sides of the heat exchanger. From the **Plot Type** drop down, you can select the X and Y axis variables for the plot. The **Tables** page displays the same information provided in the **Plots** page but in tabular form.

Figure 5.4



The type of analytical information found in operation property views is dependent on the operation type. Regardless of what the operation is, the displayed information is automatically updated as conditions change.

5.1.3 Simulation Navigator

The Simulation Navigator acts as a shortcut means of quickly viewing the property view of any stream or unit operation within your HYSYS case.

Simulation Navigator button

You can access the Simulation Navigator by selecting the **Simulation Navigator** option from the Flowsheet menu bar or by clicking the **Simulation Navigator** button in the button bar. This will open a window allowing you to view any stream or unit operation. Selecting an object from the tree list on the left will display its property view on the right just as it would appear as if accessed from the flowsheet.

Figure 5.5

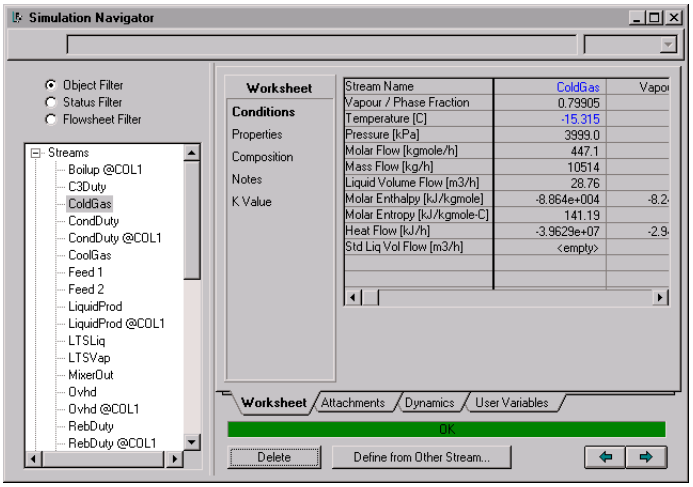


Figure 5.5 above shows a Stream property view as it would appear for the **ColdGas** Stream object currently highlighted.

There are three radio buttons that allow you to filter the flowsheet objects.

- Object - This filter organizes the attachment tree via object type. This allows you to look at just the heat transfer equipment or all of the vessels.
- Status - This filter allows you to view all of the objects via status type. There are five different status types that the objects are filtered into: OK, missing optional information, warning, missing required information and error.
- Flowsheet - All of the objects contained in a column or subflowsheet are shown under their respective column/ subflowsheet title. The objects in the main flowsheet are just listed in alphabetical order.

For certain objects. you may need to enlarge the view horizontally to see more of the property view. This can be performed by selecting a window edge with the mouse and dragging to resize the view.

5.2 Navigation

HYSYS provides two Navigational aids that essentially bring the Multi-Flowsheet architecture into a flat space. A simulation containing a Main Flowsheet and Sub-Flowsheets (Columns and/or Template SubFlowsheets) can be considered as having a Directory/File structure. Flowsheets are directories (with the Main Flowsheet being the Root directory) with Flowsheet elements (streams, operations or utilities) being the files. However, one difference is that Sub-Flowsheets are both Flowsheet Elements (within the main simulation) as well as Flowsheets themselves. The effect of this difference on how you use the navigational tools will be illustrated in the following subsections.

The two Navigators provided are:

Navigator	Definition
Object Navigator	Locate and view any Flowsheet Element within any Flowsheet, or enter the build environment for a Flowsheet. The object navigator is also used to select objects for Utilities.
Variable Navigator	Select a process variable for an object from any Flowsheet. You will use this Navigator during the installation of Controllers, Adjusts, Strip Charts, Spreadsheets, etc.

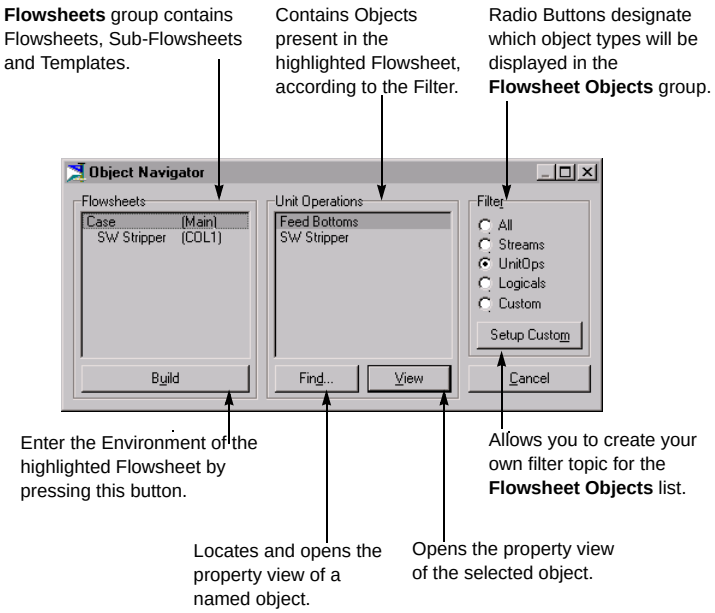
5.2.1 Object Navigator

Navigator Button

You can access the **Object Navigator** (shown in Figure 5.6) in four ways:

- Double click on an empty area of the DeskTop.
- Press the ***Navigator*** button on the Button Bar.
- Choose the **Flowsheet** option from the Menu Bar, and then choose **Find Object**.
- Press **F3.**

Figure 5.6



Object Filter

The **Object Filter** group is common to both the Object and Variable Navigators. These radio buttons allow you to filter the **Object** list to only those of a certain type. For example, choosing the **Streams** radio button will show only streams.

The ***Setup Custom*** button allows you to define your own filter criteria:

When a '+' is located after a name in the list, highlight the name and press ENTER or double click on the name to expand the list.

1. Access the **Select Type** list of available objects by selecting the **Custom** radio button or the ***Setup Custom*** button. The ***Setup Custom*** button is used when the **Custom** radio button is already selected.

2. From the **Select Type** dialog, choose the stream type or unit operation type and press ***OK***.

Locating a Property View

The Object Navigator works in a left to right sequence. When you highlight a Flowsheet, all Flowsheet objects (based on the current filter) are displayed in the **Flowsheet Objects** group.

To access a property view for a specific Flowsheet object, use one of the following methods. For all three of the methods, you will need to first select the appropriate Flowsheet from the **Flowsheets** group.

- Highlight the desired object in the **Flowsheet Objects** group and press the ***View*** button.
- Double click on the object in the **Flowsheet Objects** group.
- Press the ***Find*** button. This produces a dialog in which you provide the name of the Object you wish to locate.

When you have located the desired object, the Navigator closes and places you in the Property View for that stream or operation.

In the **Object Navigator** view (shown in Figure 5.6), **SW Stripper** appears as both a Flowsheet and a Flowsheet Object. To the main simulation, each Sub-Flowsheet appears as a single operation and has an appropriate property view (or outside view). Selecting **SW Stripper** from the **Flowsheet Objects** group brings you to this outside view.

Entering Build Environments

You can enter the **Build Environment** for any Flowsheet directly from the Object Navigator. Highlight the Flowsheet (in the **Flowsheets** group) and select the ***Build*** button, or double click on the Flowsheet directly.

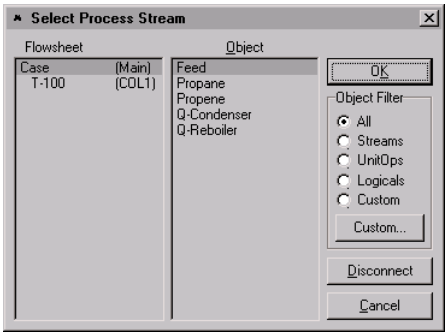
Selecting an Object for a Utility

Refer to Chapter 8 - Utilities for more information concerning available utilities.

A simplified version of the Object Navigator is used when an object must be selected for a utility. The same functionality is present, in that a Flowsheet must be selected and then an object from within that Flowsheet highlighted. However, you cannot enter other build environments or access the property views of objects from this form of the Navigator. Its sole purpose is to present the available objects for the utility.

HYSYS will automatically filter the list of objects according to the type of utility. For instance, an Envelope utility performs calculations for streams, thus the Navigator will show only available streams. The Navigator accessed via the Envelope utility is shown below.

Figure 5.7



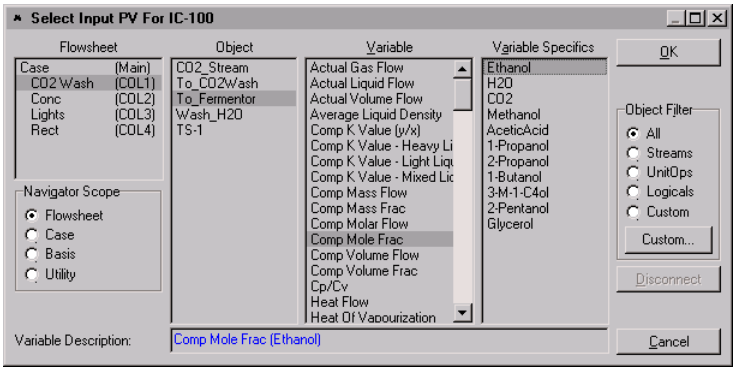
Once an object is selected for a utility, you can remove it using the Navigator. Simply re-access the Navigator and press the ***Disconnect*** button.

5.2.2 Variable Navigator

When you are browsing for variables (i.e. selecting a process variable for a controller or a Strip Chart), the Variable Navigator is used. The Variable Navigator uses a similar structure to the Object Navigator, but allows for a more detailed search. When the **Flowsheet** radio button in the **Navigator Scope** group box is selected, you can select the **Flowsheet**, the **Object**, the **Variable**, and a **Variable Specific** simultaneously.

For example, consider the situation where you want to select the Process Variable for a controller. When you press the ***Select PV*** button (from the **Controller Connections** page), the Variable Navigator will appear.

Figure 5.8



Using the Variable Navigator

Refer to Section 5.2.1 - Object Navigator for details on list filtering.

When selecting a variable, you work your way from left to right through the groups. The Object Filter eliminates selections from the **Object** group according to your selection of a radio button. Selected items will be highlighted, as shown in the previous **Variable Navigator** view.

The procedure used for selecting a process variable is:

1. Choose the **Flowsheet** radio button from the **Navigator Scope** group box.
2. From the **Flowsheet** group, select the Flowsheet from which the process variable will be selected, in this case, **CO2 Wash**.
3. The **Object** group contains all the objects (streams, operations, and logicals) within the chosen Flowsheet. Select **To_Fermentor** to be the process variable.
4. In the **Variable** group, all the variables relevant to the chosen Object are displayed. In this case, **Comp Mole Frac** is selected.
5. For certain variables (such as component specific variables), the available **Variable Specifics** are listed in the fourth column. Choose **Ethanol**.
6. The **Variable Description** box contains the description that will appear in the Variable cell for a controller, and the label for the Variable in a Strip Chart. You can change the name supplied by HYSYS.
7. To complete the selection, press the ***OK*** button.

Below the **Object Filter** group, there are two buttons, ***Disconnect*** and ***Cancel***. The ***Disconnect*** button removes a connected variable while the ***Cancel*** button simply closes the Navigator without saving any changes.

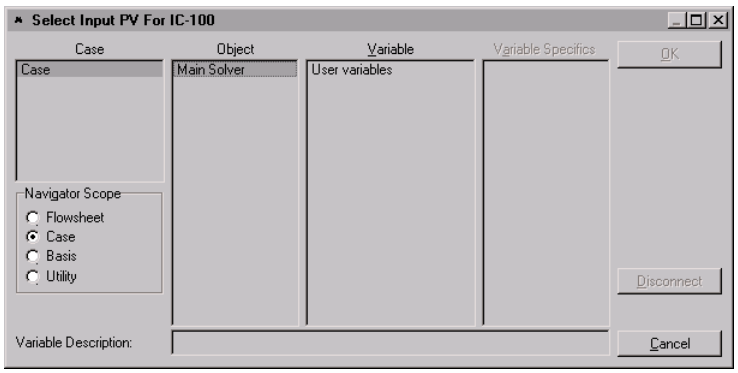
Navigator Scope

The final section of the Variable Navigator is the **Navigator Scope** group. This group contains four radio buttons.

*Column Spec Values will appear as **Variable Specifics** when you select **Spec Value** as the **Variable**.*

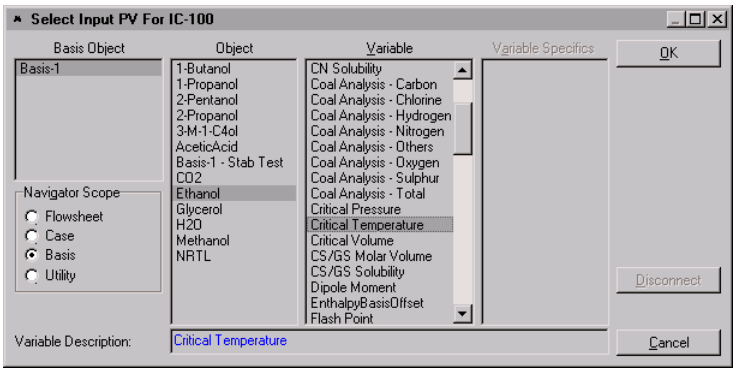
Object	Definition
Flowsheet	When this radio button is selected, the Navigator displays the previous view. The **Object** group box contains a list of all the Flowsheet Objects, as determined by the Object Filter.
Case	This radio button is primarily used to access general case information; for instance, information concerning the Main Solver or Optimizer. You can also use this radio button when a column is the Main Flowsheet, i.e. the case is a column template, to access column specification variables.

Figure 5.9



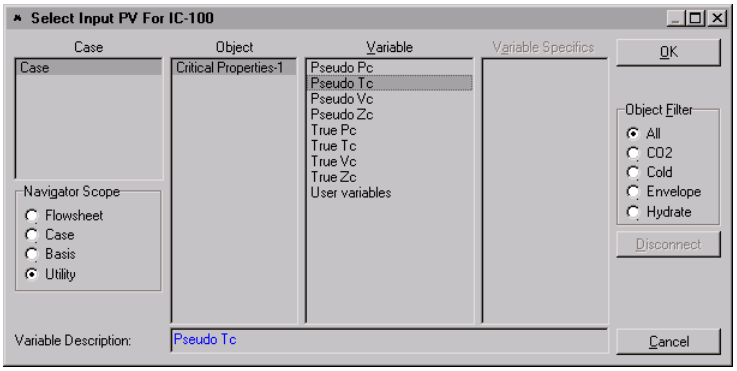
Object	Definition
Basis	When this radio button is selected, the Navigator shown in **Figure 5.10** appears. If different property packages or components are being used for SubFlowsheets, they can be accessed by using this radio button. In the **Basis Object** group box, each basis for the simulation will be listed. Once a basis is highlighted, its components and property package will be shown in the **Object** group box. Each component will have a list of variables from which to choose. For the property package selection in the **Object** group, you will be able to select options from the **Variable** group, the **Variable Specifics** group and the **More Specific** group.

Figure 5.10



Object	Definition
Utilities	When the **Utility** radio button is selected, the Navigator shown below appears. A special utility **Object Filter** appears, from which the **Object** list can be shortened. The Object Filter **All** radio button shows all of the Utilities installed in the case. The **Variables** group displays only the variables associated with the chosen Utility. For example, the variables for an **Envelope** utility are shown here.

Figure 5.11



5.3 DataBook

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

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

Databook Features
Process Data Tables
Strip Charts: • **Historical Data Table** • **Current Values Table**
Data Recorder Scenarios
Case Studies
Plotting for the Data Recorder and Case Study tool

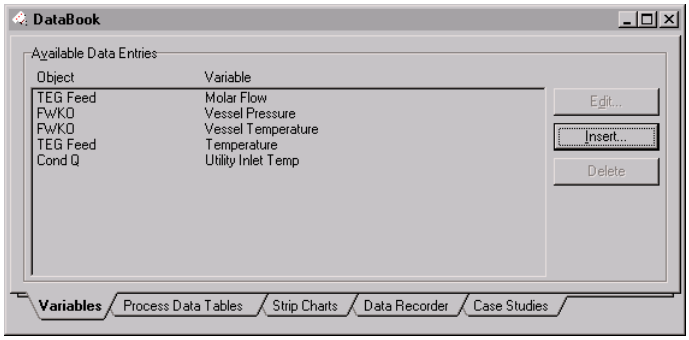
You can access the DataBook by using the CTRL D hot key combination, or via the **Tools** option on the Menu Bar.

5.3.1 Variables Tab

*You can place as many variables as you want on the **Variables** tab.*

All variables that will be used for the features of the DataBook are added on the **Variables** tab. The variables added to the DataBook are displayed in the **Available Data Entries** group box. For each variable, the Object to which it is attached and a Variable description are listed.

Figure 5.12



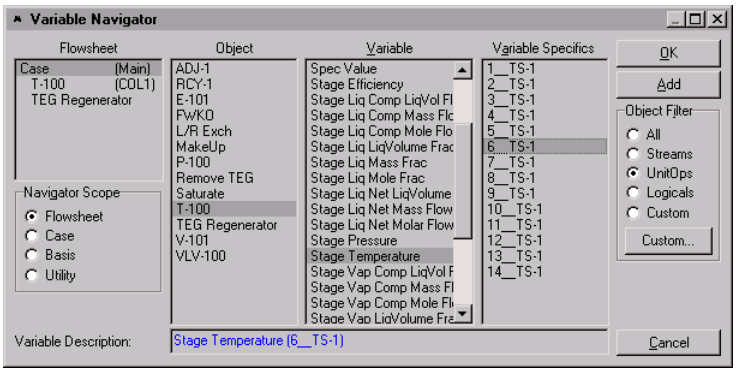
Adding a Variable

Refer to Section 5.2.2 - Variable Navigator for instructions on the use of the Variable Navigator.

The procedure for adding a variable is illustrated with the following example. The variable to be added is the tray 6 temperature in a Column operation.

1. Select the ***Insert*** button from the **Variables** page to open the **Variable Navigator**.

Figure 5.13



2. From the **Flowsheet** group, select the Main Flowsheet containing the variable.
3. Move to the **Object** group and select **T-100**.
4. In the **Variable** group, select **Stage Temperature**.
5. More detail is needed to fully define the Variable. Move to the **Variable Specifics** group and select **6_Main TS**.
6. The **Variable Description** box contains the name that will be associated with the variable. If you want you can change the name at this time. In this example, the name has been changed to **Stg6_Temp**.
7. Press the ***OK*** button to return to the **Variables** tab of the DataBook.

*Notice that the **Object Filter** has been used to display only **Unit Ops**.*

When you return to the DataBook, note that **T-100** is shown under **Object** and **Stg6_Temp** is under **Variable**.

Editing a Variable

To edit a variable, highlight it in the **Available Data Entries** group and press the ***Edit*** button. This will produce the **Variable Navigator** view. The Navigator selections are highlighted and will correspond to the selected variable. You can change any of the selections: **Flowsheet**, **Object**, **Variable**, **Variable Specifics** or the **Description**. After making changes, press the ***OK*** button. If you decide after making the changes that you do not want them, press the ***Cancel*** button. You will return to the **Variables** page and your original variable selections will be intact.

Deleting Variables

*When a Variable is deleted from the **Variables** page, it is removed from all features in the DataBook, i.e. the Process Data Tables, the Strip Charts, etc.*

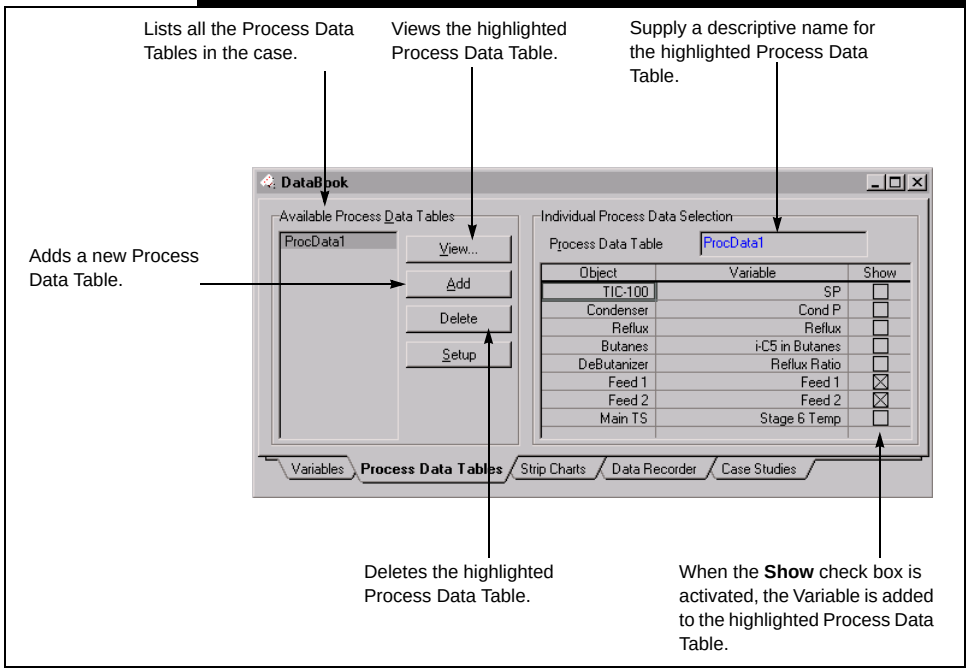
Any variable added to the DataBook can be deleted from the **Variables** page. Highlight the Variable and press the ***Delete*** button. HYSYS does not require confirmation of the deletion, so make sure you have selected the correct variable.

5.3.2 Process Data Tables Tab

*Variables for the Process Data Tables are selected on the **Variables** tab of the DataBook.*

Process Data Tables are installed individually via the **Process Data Tables** tab. On this tab, you can view, add, or delete customized process data tables. For each table, you can add any combination of key process variables from the list of available variables. Variables can be used in multiple Process Data Tables.

Figure 5.14



Adding a Process Data Table

When the **Process Data Tables** tab is first accessed, there will be no **Available Process Data Tables** in the list. Refer to Figure 5.14 while you create a Process Data Table.

Variables can be shown in more than one Process Data Table.

1. Select the ***Add*** button.
2. The default name **ProcData1** will appear in the list of available tables. If desired, enter a different name in the **Process Data Table** input cell.
3. In the **Individual Process Data Selection** group box, all variables available in the DataBook will be listed. Activate the **Show** check box for each variable that you want to appear in this Process Data Table.

Deleting A Process Data Table

When a Process Data Table is deleted, the variables associated with it are not deleted from the DataBook.

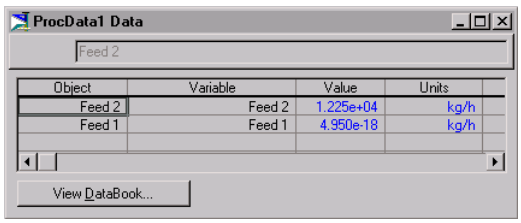
There are two methods available for deleting a Process Data Table from a case. Each of the methods is permanent, in that a deleted Process Data Table cannot be recovered. Also, HYSYS does not ask you for confirmation before deleting a Process Data Table. The methods available are:

- Highlight the Process Data Table name in the **Available Process Data Tables** group and press the ***Delete*** button.
- Highlight the Process Data Table name and press the **DELETE** key on the keyboard.

Viewing A Process Data Table

To view existing Process Data Tables use one of the following methods:

Figure 5.15



- Highlight the Process Data Table name in the **Available Process Data Tables** group and press the ***View*** button.
- Double click on the name of the Process Data Table in the Available Process Data Tables.

The Process Data Table shows the following for each variable:

- The Object to which the variable is attached
- The descriptive Variable name
- The current Value
- The Units

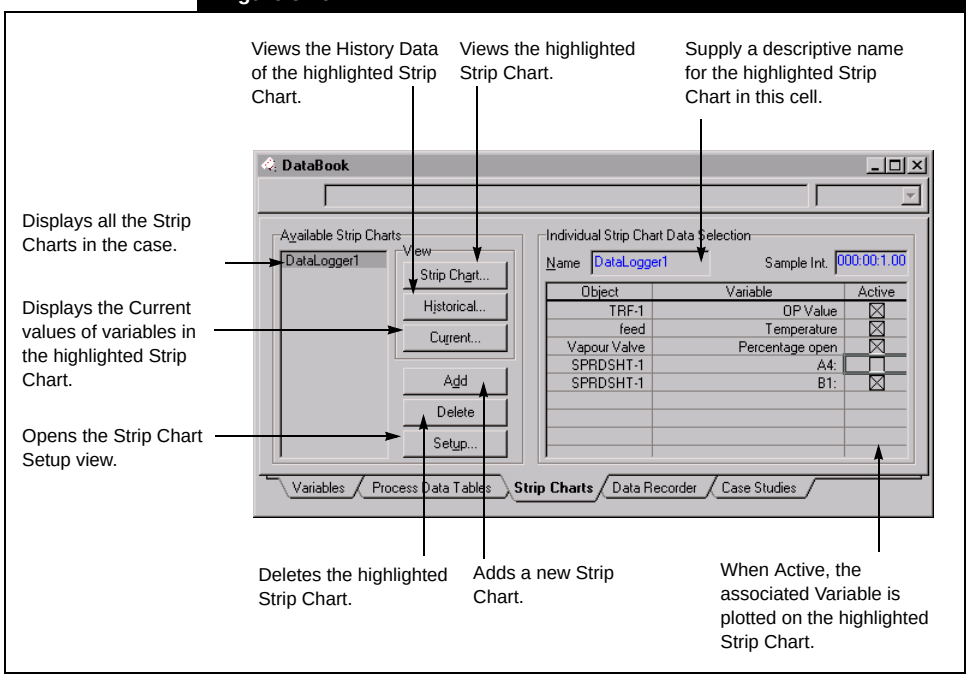
5.3.3 Strip Charts Tab

*Variables for the Strip Charts are selected on the **Variables** tab of the DataBook.*

Monitoring the response of key process variables during **dynamic** calculations is best accomplished via the **Strip Charts** tab. Strip Charts allow you to monitor the behaviour of process variables in a graphical format while calculations are proceeding. Current and Historical values for each Strip Chart are also tabulated for further examination.

Strip Charts are installed individually via the **Strip Charts** tab. Multiple Strip Charts are allowed, and each of these can have an unlimited number of variables charted. However, since the same variable can be used in more than one Strip Chart, it is recommended that you utilize multiple Strip Charts with no more than six variables per Strip Chart.

Figure 5.16



The **Strip Charts** tab is split into two groups: **Available Strip Charts** and **Individual Strip Chart Data Selection**.

Available Strip Charts Group

The **Available Strip Charts** group consists of the **Available Strip Charts** list that displays all the previously created strip charts. Six buttons border the list box which allow you to manipulate the strip charts and their properties.

Button	Description
Strip Chart	Opens the Strip Chart view for the current selection in the Available Strip Charts list.
Historical	Displays the historical data of the highlighted strip chart in tabular form.
Current	Displays the Current values of the active variables in the selected Strip Chart.
Add	Adds a new Strip Chart to the Available Strip Chart list.
Delete	Deletes the selected Strip Chart.
Setup	Opens the **Logger Setup** view. The **Logger Setup** view displays the number of samples that will be stored in the Strip Chart. The view also displays how frequently data is recorded into the strip chart.

Individual Strip Chart Data Selection Group

The **Individual Strip Chart Data Selection** group contains four objects:

Object	Description
Name	The name of the Strip Chart.
Sample Int.	The sampling interval.
Variable Selection	A matrix that displays the variables selected on the **Variables** tab. The matrix consists of three columns: • **Object** - displays the flowsheet source for the variable. • **Variable** - displays the selected variable. • **Active** - toggles the variables in and out of the Strip Chart.

Adding a Strip Chart

Refer to Figure 5.16 as you add a new Strip Chart to the DataBook.

1. Select the ***Add*** button in the **Available Strip Charts** group.
2. HYSYS installs the new Strip Chart and automatically names it. In this case, the name is **DataLogger1**.

Variables can be shown in more than one Strip Chart.

3. The name of the currently selected Strip Chart, **DataLogger1** also appears in the **Strip Chart Name** cell. You can modify the name directly in the input cell.
4. In the **Individual Strip Chart Data Selection** group box, all variables available in the DataBook will be listed. Activate the **Active** check box for each variable that you want to appear in this Strip Chart.

Deleting a Strip Chart

When a Strip Chart is deleted, its attached variables are not deleted from the DataBook.

There are a number of methods available for deleting a Strip Chart from a case. Each of the methods is permanent, in that a deleted Strip Chart cannot be recovered. Also, HYSYS does not ask you for confirmation before deleting a Strip Chart. The methods available for deleting a Strip Chart are:

- Highlight the Strip Chart name in the **Available Strip Charts** group and press the ***Delete*** button.
- Highlight the Strip Chart name and press the **DELETE** key on the keyboard.
- Highlight the Strip Chart name, Object Inspect it and select **Delete** from the menu that appears.

Viewing the Strip Chart

The most common means of monitoring the trends of targeted process variables is via the Strip Chart. This is a dynamic view that is being continually redrawn with time labels shifting.

To view existing Strip Charts use one of the following methods:

- Highlight the Strip Chart name in the **Available Strip Charts group** and press the ***View*** button.
- Double click on the name of the Strip Chart.
- Highlight the name of the Strip Chart, and object inspect it. From the menu, choose **View** to display the Strip Chart.

If the Integrator has been started, the Strip Chart begins to accumulate data points as soon as it is created.

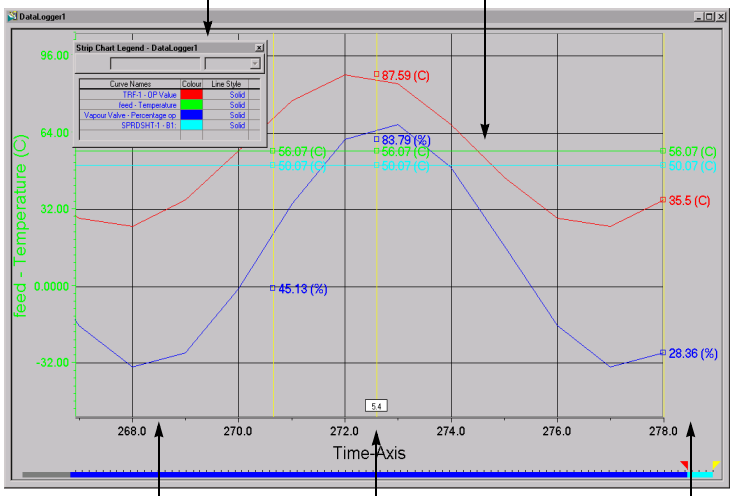
Strip Chart View

Figure 5.17

The Legend shows the Names and Formats of the Variables, as they are shown on the Strip Chart.

By default like variables are grouped under a similar axis range. You may however create you own y axis in the **Graph Controls** property view.

Strip Chart variables are grouped into their unit sets. Grouped variables share the same y-axis range.



Time is always on the x-axis.

Interval Markers are used to measure variables at certain instances in the strip chart.

Current values of the Strip Chart variables.

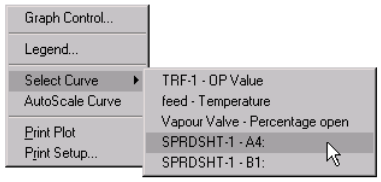
You can manipulate the information displayed on the strip chart within the **Strip Chart** view. There are several functions you can perform directly on the strip chart.

Selecting Curves

The y-axis will display the range and units of a specific variable in the strip chart depending on which variable you select in the strip chart. You may select a curve in one of two ways:

- Click any part of the strip chart variable curve on the **Strip Chart** using the primary mouse button.
- Object inspect the **Strip Chart**. Choose **Select Curve** from the Object Inspection menu and select strip chart variable.

Figure 5.18



Manipulating the Y-Axis Range

*If you select the background of the strip chart the following mouse cursor will appear. It will allow you to move in **any direction** along the strip chart.*

*If you simply wish to move **vertically** across the strip chart select a curve with the primary mouse button and the following mouse cursor will appear.*

You can manipulate the y-axis range of the strip chart. In order to modify the range of what is displayed:

1. Click and hold any part of the strip chart. A multi-directional cursor will appear, as shown in the sidebar.
2. Drag the **Strip Chart** up if you wish to display a lower range of values on the y-axis. Drag the **Strip Chart** down if you wish to display a higher range of values.

By default, **Strip Chart** curves are grouped into their unit sets. For instance, all temperature variables are associated and displayed with the same y-axis range and units. By manipulating the range of a temperature variable in the strip chart, you change the range of all temperature variables associated with that axis.

If you wish to associate a different range to a variable in the **Strip Chart**, you must first create your own axis. You may create different axes in the **Axis** tab of the **Strip Chart Configuration** view. This view can be accessed by object inspecting the **Strip Chart** and selecting **Graph Control**.

Manipulating the X-Axis Range

You may scroll across the Strip Chart by either:

- *the x-axis, or*
- *selecting the light blue section of the Log Controller Bar with the mouse cursor.*

The following cursor should appear.

You may move the cursor along the bar to the interval you wish to view.

You can manipulate the range of sampled data displayed in the strip chart. HYSYS displays, below the x-axis, a set of colours indicating what range of sampled data is displayed in the strip chart. You may increase or decrease the range of sampled data. You may scroll the strip chart over a range of recorded strip chart data. Depending on how much data is already recorded in the **Strip Chart** view, a Log Controller bar similar to one shown in Figure 5.19, will be displayed below the x-axis.

Figure 5.19

Each colour in the Log Controller Bar is described as follows from left to right:

Colour	Definition
Gray Bar	The gray section of the strip chart displays where there is no data in the strip chart.
Dark Blue Bar	The Dark Blue section shows where strip chart data is recorded.
Red Marker	The Red marker labels where data are first displayed in the strip chart. You can expand the range of display in the strip chart by “dragging” the red marker to the left (away from the yellow marker). Decrease the displayed range of data by dragging the red marker right (towards the yellow marker).
Light Blue Bar	This section graphically shows which section of the strip chart is displayed. You may click and drag the light blue strip of the Log Controller bar across the strip chart x-axis to view the history of the strip chart. Drag the light blue strip of the Log Controller bar to view the most recent recorded data.
Yellow Marker	The Yellow marker labels where the displayed data ends. You can expand the range of displayed in the strip chart by “dragging” the yellow marker to the right (away from the red marker). Decrease the displayed range of data by dragging the yellow marker left (towards the red marker).

Creating Interval Markers

You may create and display interval markers on the strip chart. Interval markers are used to measure variables at certain instances in the strip chart. The strip chart variable value appears next to where the interval marker intersects the strip chart variable curve.

You may add up to four interval markers to the strip chart. To add a new interval marker, use the following procedure:

1. Ensure that the most recent strip chart data is displayed on the **Strip Chart** view. (The light blue part of the Log Controller Bar should be located at the far right of the x-axis.)
2. Place the cursor on the right edge of the strip chart. A left arrow should be in place of the cursor.
3. Press the primary mouse button and drag the interval marker across the strip chart.

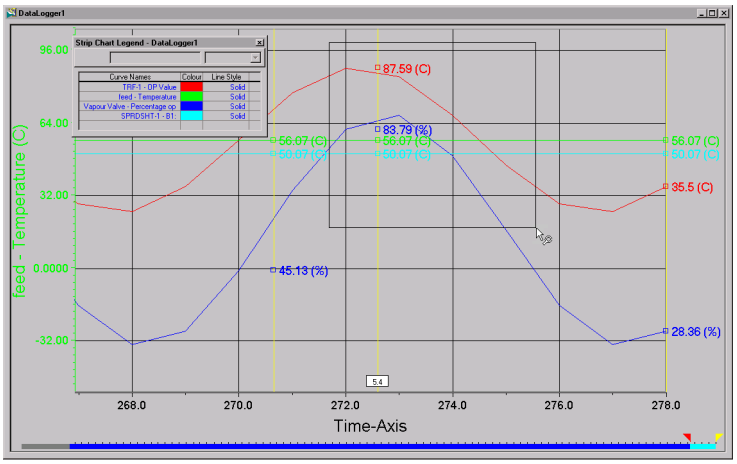
Left Arrow Cursor

Zooming in on the Strip Chart

Zoom Strip Chart Cursor

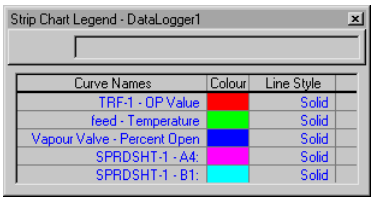
There may be instances when you want to focus on a particular area in the Strip Chart. In order to zoom in on an area of the strip chart, press the SHIFT key and click, the mouse cursor should change to a magnification glass. Drag an area onto the strip chart. The y-axes scaling will change to reflect the zoom.

Figure 5.20



Object Inspection Options

The options available through the object inspection of the Strip Chart are displayed in the following table. The **Object Inspection** menu appears if you click on the **Strip Chart** using the secondary mouse button.

Option	Description
Graph Control	The **Graph Control** option allows you to customize the appearance of the Strip Chart. You are able to modify curve and axis parameters in this option. For more information regarding the **Graph Control** option, refer to the **Graph Control** that appears later in this section.
Legend	Select the **Legend** option in order to display the **Legend** view.  The **Legend** view displays all the **Curve Names, Colours**, and **Line Styles** associated with the Strip Chart variables. You may modify the **Curve Name** or **Line Style** of any variable by clicking on the corresponding cell in the Legend view. Modify the **Curve Name** in the Edit Bar. You can choose a different **Line Style** from the Edit Bar drop down list.
Select Curve	The y-axis will display the range and units of a selected variable in the strip chart. You may select a curve in one by choosing the **Select Curve** option from the Object Inspection menu and select any strip chart variable.
Auto Scale Curve	You may choose the **Auto Scale Curve** option if you wish to have HYSYS automatically set the bounds for the y-axis of the selected curve.
Print Plot	You may print the **Strip Chart** by selecting the **Print Plot** option. HYSYS prints the strip chart as it appears on the screen.
Print Setup	Select the **Print Setup** option in order to modify any print options associated with printing the **Strip Chart**.

Graph Control

You can modify many of the strip chart characteristics in the **Strip Chart Configuration** view. You can access this view by selecting **Graph Control** in the **Object Inspection** menu. The **Strip Chart Configuration** view contains **four** tabs: **General**, **Curves**, **Axes**, and **Time Axis**.

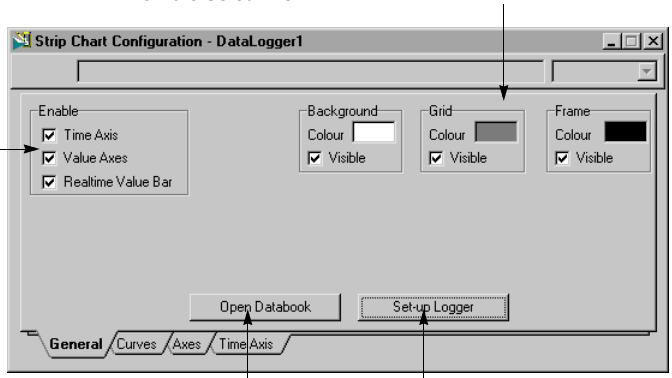
General Tab

The **General** tab contains information used to format the chart appearance.

Figure 5.21

Check the **Visible** check box in order to make the **Background**, **Grid**, or **Frame** colours visible. You may choose a colour for the item by double-clicking on the **Colour** box and selecting a colour from the **Colour** view.

Check boxes are used to modify the appearance of the Strip Chart. Activate the appropriate check box to display the option on the Strip Chart.



Click the ***Open Databook*** button in order to bring up the **Databook** view.

Click the ***Setup Logger*** button in order to bring up the **Setup Logger** view.

Curves Tab

The **Curves** tab displays information regarding the appearance of individual curves in the Strip Chart. You may also modify how strip chart variables, variable titles and units are displayed on this tab.

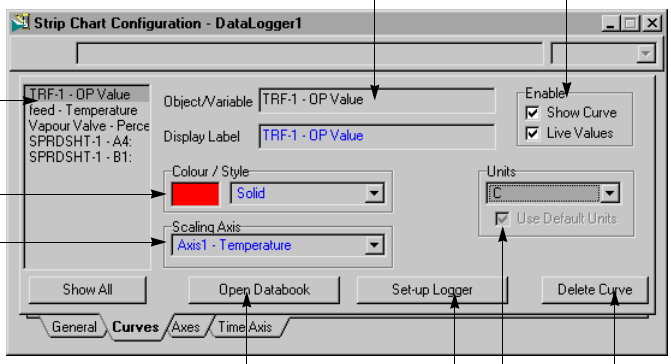
Figure 5.22

Select a specific Strip Chart variable whose plot characteristics you wish to display in the **Curves** tab. The **OP value** of operation TRF-1 is currently displayed in the **Curves** tab.

Modify the curve colour and line style in the Edit box. You may choose a colour for the item by double-clicking on the **Colour** box and selecting a colour from the **Colour** view

The **Object/Variable** and **Display Label** fields display the Strip Chart curve names. By default, the **Display Label** is copied from the **Object/ Variable** field. You may what is actually displayed in the Strip Chart by modifying the **Display Label** field.

Click the **Show Curve** check box to display or hide the selected curve. Click the **Live Values** check box to display the most recent calculated strip chart values.



Displays the Axis associated with the selected Strip Chart variable. You may

Click the ***Open Databook*** button in order to bring up the **Databook** view.

Click the ***Setup Logger*** button in order to bring up the **Setup Logger** view.

Deletes the Curve

Displays the Units of the curve.

Axes Tab

You may create, modify and delete y-axes in the **Axes** tab. The **Axes** tab allows you to create different types of axes that can be associated with different strip chart variables.

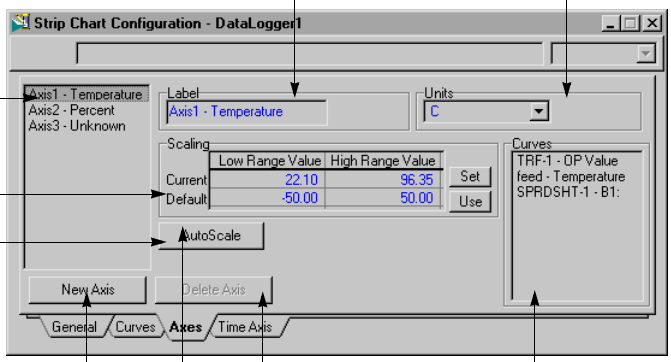
Figure 5.23

The **Object/Variable** and **Display Label** fields display the Strip Chart curve names. By default, the **Display Label** is copied from the **Object/Variable** field. You may change what is actually displayed in the Strip Chart by modifying the **Display Label** field. If you to wish to label your axis, deactivate the **Show** check box

You may label the axis with units. If you do not wish to label your axis with units, deactivate the **Show** check box.

Select a specific axis whose characteristics you wish to display in the **Axes** tab.

Modify the axis range in the **Scaling** group box. The **Current** range displays the actual range of the axis. Press the ***Set*** button to copy current ranges into the default field. Press the ***Use*** button to copy default ranges into the Current field.



Press the ***New Axis*** button to create an new axis.

Press the ***AutoScale*** button to automatically scale

Press the ***Delete Axis*** button to delete an axis you created. You cannot delete the default axes.

The Curves group box displays all strip chart variables associated with the selected axis.

Time may be entered in the format:

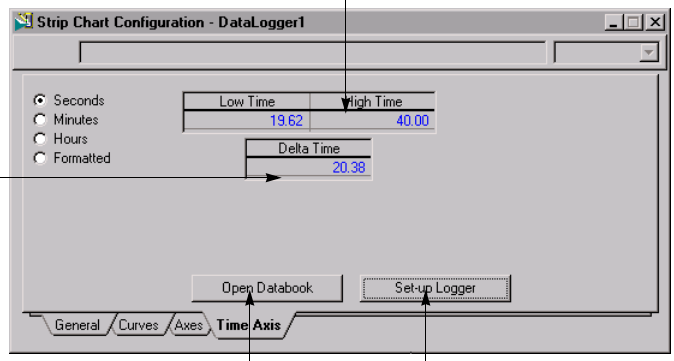
HHH:MM:SS.S

Time Axis Tab

The **Time Axis** tab is used to modify the display range of the strip chart.

Figure 5.24

The **Low Time** and **High Time** fields indicate what range of time is displayed on the strip chart. You may modify these fields directly in the **Time Axis** tab.



The **Delta Time** field represents the difference between the **High** and **Low Times**. You may modify this field.

Click the ***Open Databook*** button in order to bring up the **Databook** view.

Click the ***Setup Logger*** button in order to bring up the **Setup Logger** view.

Historical Data

*The **Current** button on the DataBook **Strip Charts** page displays the current values of the variables for the selected Strip Chart.*

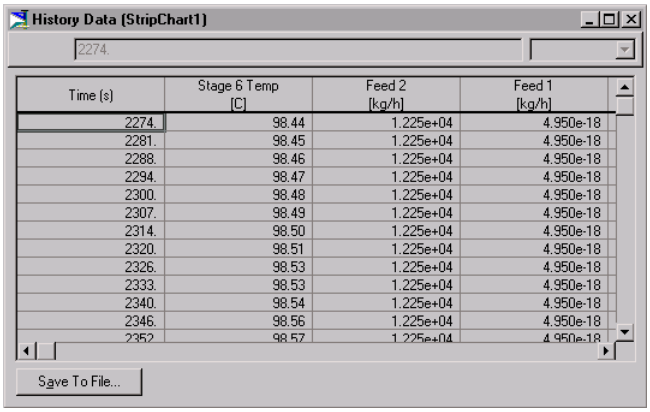
The **Historical Data** view records the data history for the variables on a Strip Chart. The number of points recorded and the time between points is determined by the **Accumulated Points** and **Sampling Rate** in the **Strip Chart SetUp** view. The Historical can only be accessed on the **Strip Charts** tab in the DataBook (by pressing the ***Historical*** button).

*The **Historical Data** view is resizable.*

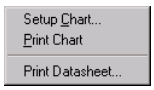
The **Historical Data** view is a scrolling view of all recorded values for each of the Strip Chart Variables. You can move up and down through the list of data. You can also print the contents of the Historical to a file with the ***Save To File*** button. This prints the set of information in the Historical to an ASCII file.

The same variable on multiple Strip Charts will be recorded independently using the settings for the Strip Chart.

Figure 5.25



Object Inspection of Strip Charts



Like most objects in HYSYS, Strip Charts can be Object Inspected. Place the mouse pointer on an area of the Strip Chart and press the secondary mouse button. The options from the menu are:

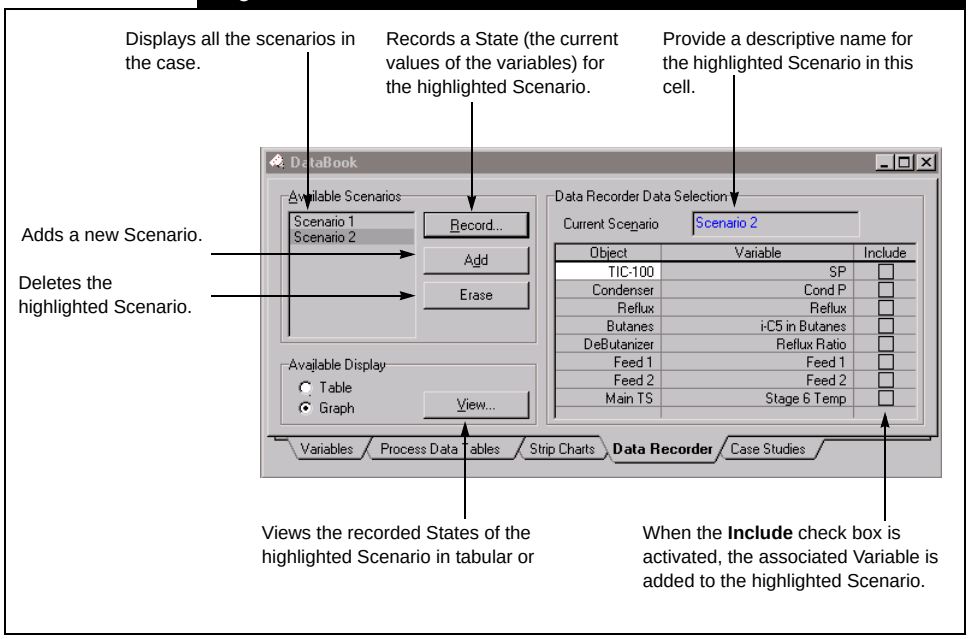
Object	Description
Setup Chart	Opens the Strip Chart SetUp view.
Print Chart	Prints the Strip Chart.
Print Datasheet	Previews or prints a Datasheet containing the Strip Chart Variables and Historical data.

5.3.4 Data Recorder Tab

*Variables for the Data Recorder are selected on the **Variables** page of the DataBook.*

The **Data Recorder** tab lets you store snap shots of your process by grouping key process variables into different **Scenarios**. You can manipulate the process in the current case and then have HYSYS record the results for the variables that you are monitoring. Each Scenario that you create on the **Data Recorder** tab can have an unlimited number of snap shots, which are called **States**. Examine the states of a Scenario in a table or on a plot.

Figure 5.26



Adding a Scenario

On the **Data Recorder** tab, you can create an unlimited number of Scenarios for the current case. Refer to the previous view as you create a Scenario:

1. Press the ***Add*** button in the **Available Scenarios** group box.
2. HYSYS automatically names the Scenario with the format Scenario#, where # represents a positive integer value greater than zero.

Variables can be included in more than one Scenario.

3. In the **Data Recorder Data Selection** group, you can provide a descriptive name for the Scenario in the **Current Scenario** input cell.

4. From the list of variable, activate the **Include** check box for each process variable which will be added to the Scenario.

Recording States

Each time a change has been made to your process, you may wish to record the key variable values which are contained in the Scenarios of the Data Recorder. To record a State, follow this procedure:

1. In the **Available Scenarios** group, highlight a Scenario.
2. Press the ***Record*** button. The **New Solved State** dialog appears. The initial default name for the new State is **State1**. Each time the ***Record*** button is pressed, HYSYS incrementally increases the integer value in the State name.

Figure 5.27

3. You can modify the State name directly in the **Name for New State** input cell.
4. Press the ***OK*** button. The key process variables of the highlighted Scenario are recorded.
5. Repeat steps #1 through #4 each time you want to record the process variables in the Scenario.

If you add a variable to a Scenario after you have recorded States, values for the new variable will not be shown in the previously recorded States.

You can record an unlimited number of States for a given Scenario.

Viewing a Scenario in the Data Recorder Notebook

*If more than one Scenario has been created, the Scenarios will be grouped in a **Notebook** format.*

There are two options available when you want to view the States contained within a Scenario. The States can be displayed in tabular or graphical format.

1. Access the **Data Recorder** tab of the DataBook.
2. Highlight a Scenario in the **Available Scenarios** group.
3. In the **Available Display** group, select either the **Table** or the **Graph** radio button.

4. Press the ***View*** button to access the tab for the selected Scenario in the **Data Recorder** Notebook. Depending on the radio button selection (step #3), either a table or a plot will appear.

Figure 5.29

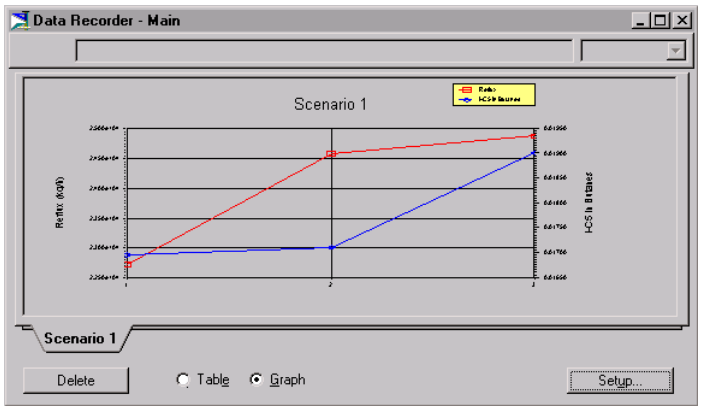
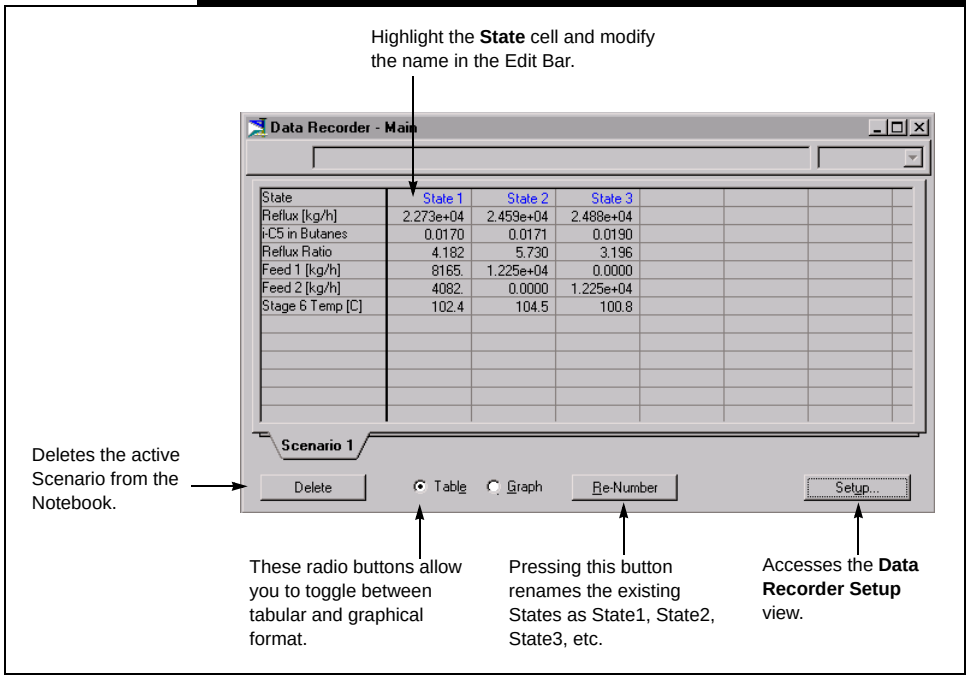


Figure 5.28



You can customize the Scenario plot by accessing the Graph Control property view. Object inspect anywhere on the plot area and select **Graph Control** from the menu.

Refer to Section 6.4 - Graph Control for information on customizing plots.

If more than 2 variables are **Active** on the **Data Recorder Setup** view, only the first 2 will be plotted.

Deleting a Scenario

There are a number of methods available for deleting a Scenario from a case. Each of the methods is permanent, in that a deleted Scenario cannot be recovered. Also, HYSYS does not ask you for confirmation before deleting a Scenario. The methods available for deleting a Scenario are:

- Highlight the Scenario name in the Available Scenarios group and press the ***Erase*** button.
- Highlight the Scenario name and press the **DELETE** key on the keyboard.
- Open the Data Recorder Notebook, select a page tab for a Scenario and press the ***Delete*** button.

*Refer to the **Viewing a Scenario** section for information concerning the **Data Recorder Notebook**.*

Deleting States

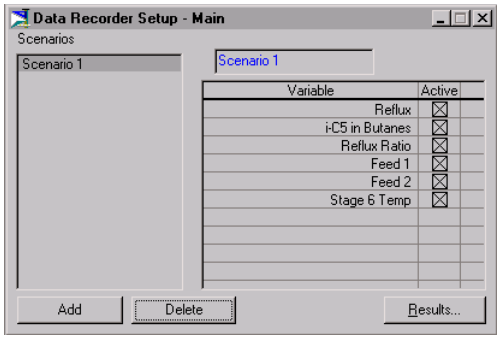
To remove a recorded state from a Scenario, you must access the page tab for the Scenario in the **Data Recorder Notebook**. Switch to tabular format by selecting the **Table** radio button and then use one of the following methods to delete the recorded state:

- Highlight the State and press the **DELETE** key.
- Object inspect any cell for the State and select Delete from the menu.

Data Recorder Setup

The **Data Recorder Setup** view provides a list of all Scenarios installed in the DataBook**.** From this view, you can select the variables that will appear in the table and on the **y-axis** of the plot for each Scenario. You are limited to two y-axis variables per plot.

Figure 5.30



For each Scenario, select the variables to display:

1. Highlight a Scenario from the **Scenarios** group.
2. From the list of variables, activate the **Active** check box for the variable(s) to be displayed on the plot and in the table.

The following buttons are also available on the **Data Recorder Setup** property view:

Button	Description
Add	Adds a Scenario to the DataBook.
Delete	Removes the selected Scenario from the DataBook.
Results	Opens the **Data Recorder Notebook** to the page tab of the highlighted Scenario.
Close	Closes the **Setup** view.

5.3.5 Case Studies Tab

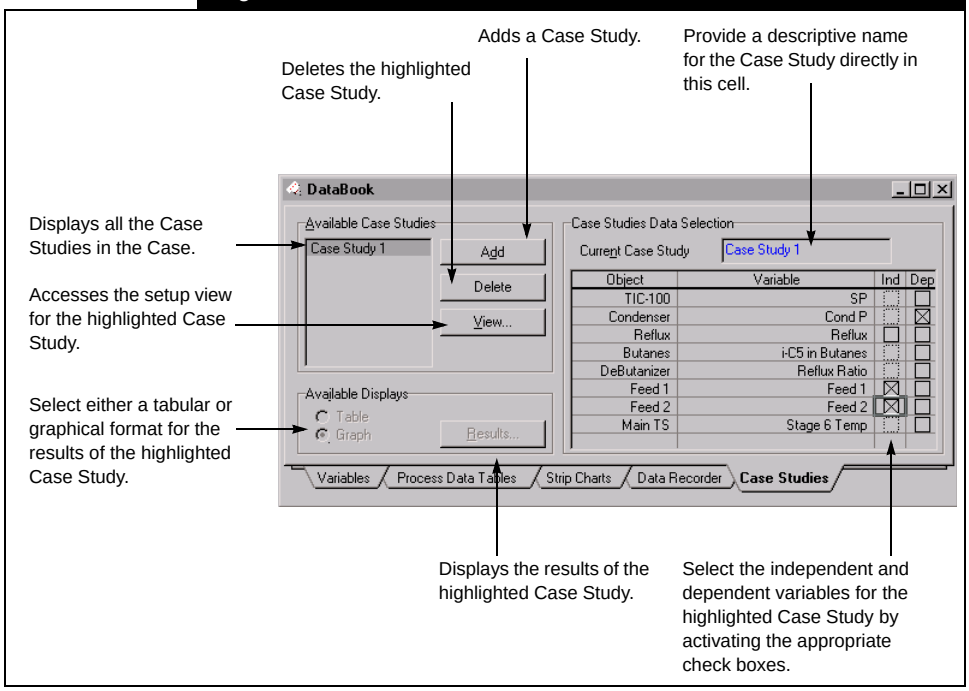
*Variables for the Case Study are selected on the **Variables** page of the DataBook.*

The case study tool allows you to monitor the steady state response of key process variables to changes in your process. From the list of variables created on the **Variables** tab, you designate the independent and dependent variables for each case study. For each independent variable, you will specify a lower and upper bound, as well as a step size. HYSYS varies the independent variables one at a time, and with each change, the dependent variables are calculated and a new State is defined. HYSYS shows the number of States which will be calculated as you define the bounds and step size of the independent variables.

Since a maximum of 2 independent variables can be plotted, you can limit the number of States and minimize solution time by selecting only two independent variables per case study.

Once the Case Study has solved, you can examine the States in a table or view the results in a plot. Although you can select as many variables as you like for a Case Study, a maximum of three variables can be shown on a plot. One independent variable and two dependent variables will produce a two-dimensional plot while two independent variables and a single dependent variable will be shown on a three-dimensional graph.

Figure 5.31



Adding A Case Study

You can create an unlimited number of Case Studies in a case. Refer to Figure 5.31 as the procedure for adding a case study is outlined:

You can use the same variables in different case studies.

1. Press the ***Add*** button in the **Available Case Studies** group.
2. HYSYS automatically names the case study with the format **Case Study #**, where # represents a positive integer above zero.
3. In the **Case Studies Data Selection** group, activate the **Ind** check box for the (independent) variables that HYSYS will vary.
4. Activate the **Dep** check box for the calculated (dependent) variables.
5. In the **Current Case Study** input cell, you can change the name of the case study.

Removing A Case Study

There are a number of methods available for deleting a Case Study from a case. Each of the methods is permanent, in that a deleted Case Study cannot be recovered. Also, HYSYS does not ask you for confirmation before deleting a Case Study. The methods available for deleting a Case Study are:

*Refer to the **Viewing a Case Study in the Case Studies Notebook** section for information on the Notebook format.*

- Highlight the Case Study name in the **Available Case Studies** group and press the ***Delete*** button.
- Highlight the Case Study name and press the **DELETE** key on the keyboard.
- Open the Case Studies Notebook, select a tab for a Case Study and press the ***Delete*** button.

Case Study Setup

Access the **Case Studies Setup** view in one of three ways:

- Press the ***View*** button on the Case Studies page of the DataBook.
- Double click on the Case Study name in the **Available Cases Studies** group.
- Press the ***Setup*** button from the Case Studies Notebook.

Three tabs are available on which you can specify the parameters of each case study:

Tab	Description
Independent Variables Setup	Define the bounds and step size for each of the independent variables.
Display Properties	Select which dependent variable(s) to display in the Table and on the Graph.
Failed Status	Displays any states that have failed during the case study.

Common to both of the aforementioned tabs are the buttons which line the bottom of the **Setup** property view:

Button	Description
Add	Adds a case study to the case
Delete	Removes the highlighted case study from the case
Results	Opens the Case Studies Notebook
Start/Stop	Starts or stops the solution of the highlighted case study. The ***Stop*** button is shown only when a case study is solving.

Independent Variables Setup Tab

*Steps #1 and #2 are reversible. You could select a Case Study from the **Case Studies Setup** property view.*

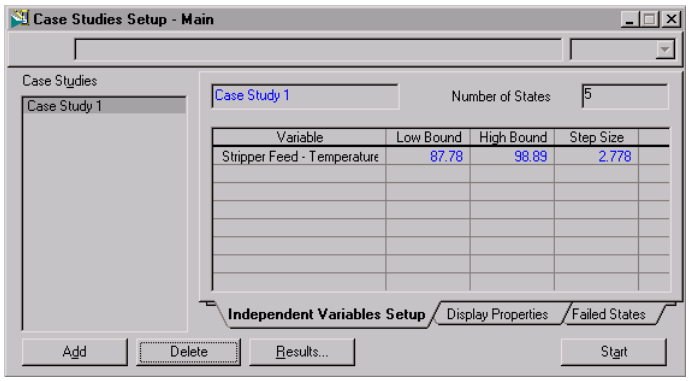
For each case study, the range and step size for the independent variables must be defined. HYSYS will automatically calculate the **Number of States** which will occur in the case study. To define the variable ranges, proceed as follows:

1. In the **Case Studies** list box in the DataBook, highlight a case study from the **Available Case Studies** group.
2. Press the ***Setup*** button to access the **Case Studies Setup** property view
3. For each independent variable, supply a **Low Bound**, **High Bound** and **Step Size**.
4. Note the **Number of States** is displayed in the upper right corner.

*A maximum of 2 independent variables can be plotted. If more than 2 are used in a case study, graphical results will **NOT** be available.*

Press the ***Start*** button to begin the calculations.

Figure 5.32



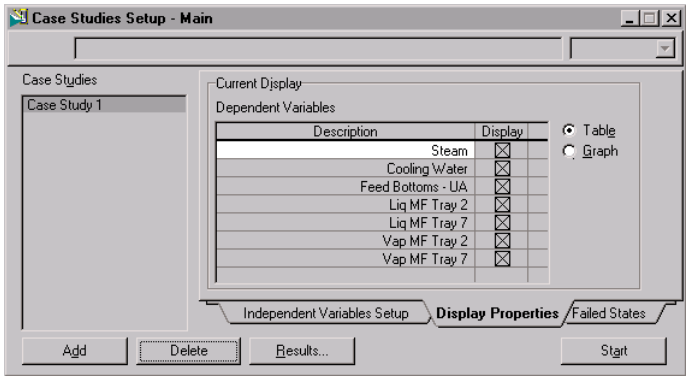
Display Properties Tab

The list of Dependent Variables will be empty until the calculations for the Case Study have been started.

On the **Display Properties** tab, you can select which dependent variables to display in the Table and on the Graph for each case study. Activate the **Display** check box for each Dependent variable that you want to display in the Results.

*If **two** independent variables are active, only the first activated **dependent** variable will be displayed on the plot. If **one** independent variable is used, the first 2 activated dependent variables will be shown on the plot.*

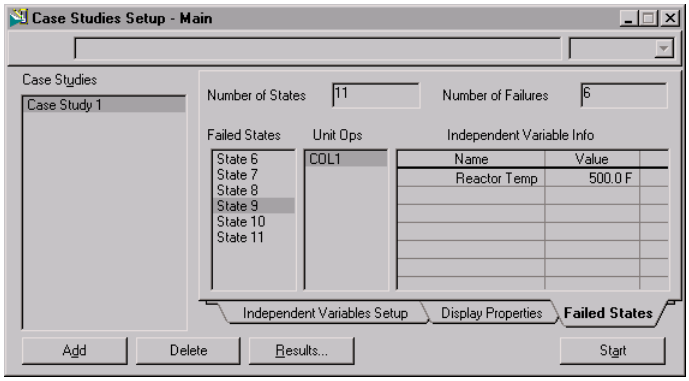
Figure 5.33



Failed States Tab

The **Failed States** tab displays the number of states that the case study consists of as well as the number of states that have failed. The **Failed States** and **Unit Ops** group lists the states that have failed and the name of the unit op where the failure occurred. The value of the independent variable for each failed state is displayed in the **Independent Variable Info** group.

Figure 5.34



Case Study Display

The Case Studies Notebook provides a tab for each Case Study and displays the results in tabular or graphical format. You can access the Case Studies Notebook from two different views:

Case Studies Tab in the DataBook

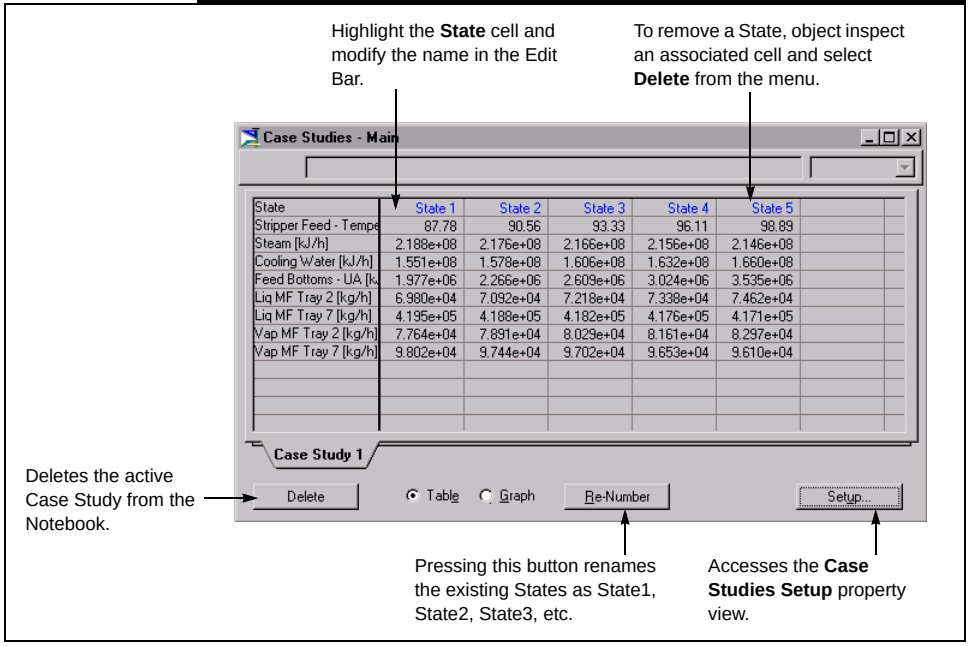
1. Highlight a Case Study in the **Available Case Studies** group.
2. Select either the **Table** or **Graph** radio button in the **Available Displays** group.
3. Press the ***Results*** button.
4. Depending on the radio button selection (step #2), either a table or a plot will be displayed for the **Case Study**.

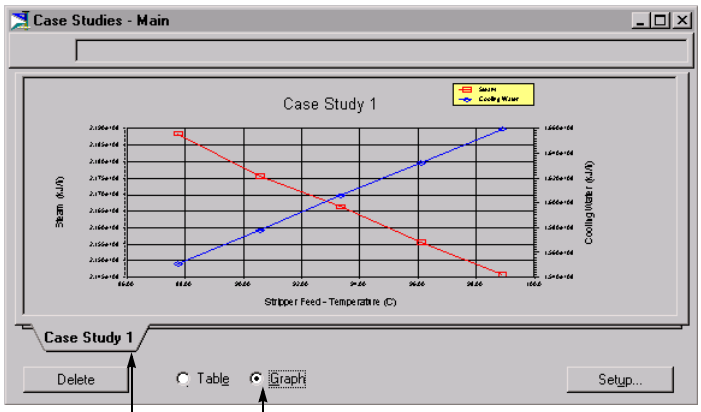
*Note that within the **Case Studies** Notebook, you can use the radio buttons to toggle between tabular and graphical format.*

Case Studies Setup View

1. From the **Case Studies Setup** view, select either the **Table** or **Graph** radio button.
2. Press the ***Results*** button.
3. Depending on the radio button selection (Step #1), either a table or a plot will be displayed for the **Case Study**.

Figure 5.35



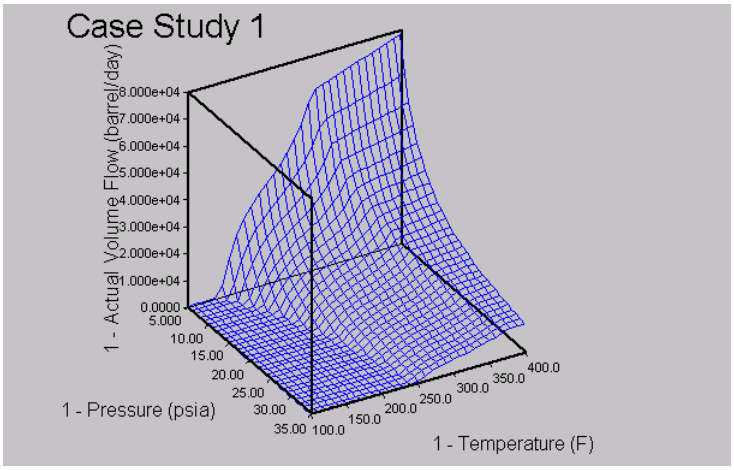


Refer to ***Section 6.4 - Graph Control*** *for information on customizing plots.*

You can customize the Scenario plot by accessing the **Graph Control** property view. Object inspect anywhere on the plot area and select **Graph Control** from the menu.

Multi-Dimensional Graphing

Figure 5.36



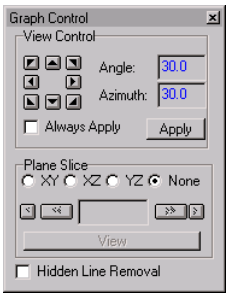
Often, you may be conducting Case Studies that involve more than one independent variables. HYSYS allows you to not only view the results of the study in table form, but to also plot them in a 3-dimensional graphing environment.

HYSYS will automatically plot three dimensionally if you run the Case Study with two independent variables selected. To view the three dimensional plot simply select the **Graph** radio button. Please note that the limits of the system allow for only three dimensions. Any more than two independent variables will result in no graph being produced.

3-Dimensional Graph Control

To access the Graph Control click the right mouse button with the cursor over the plot and select **Graph Control**. The **Graph Control** view will appear as shown:

Figure 5.37

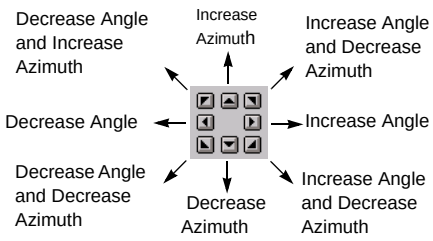


The **Azimuth** field defines the angle between a horizontal plane and the XY plane of the plot. This means that an azimuth of 0.0 will result in a view of the XY plane as a single horizontal line across the screen. The Azimuth may be manipulated by selecting the **Azimuth** field, entering a value, and then pressing the ***Apply*** button.

The **Angle** field defines the angle between the vertical and the **XZ** (**YZ**) plane of the plot. Increasing the angle will cause the graphic to rotate counter-clockwise. The angle may be manipulated by selecting the **Angle** field, entering a value, and pressing the ***Apply*** button.

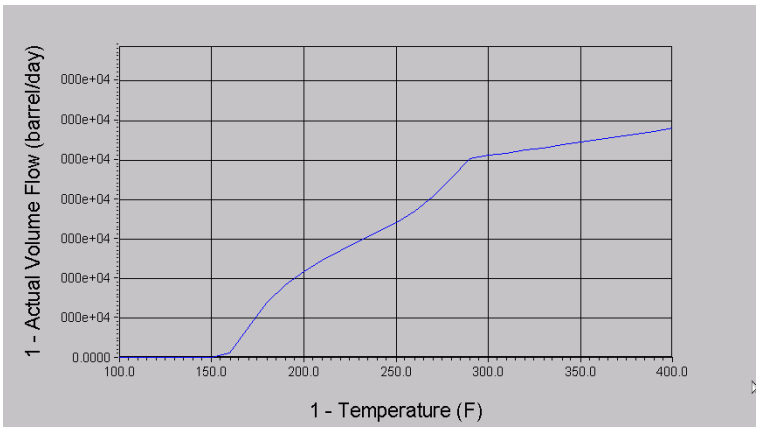
Alternatively, the angle and azimuth can be manipulated by pressing the arrow buttons of the keypad located in the **View Contro**l group. The action of each arrow button is noted as shown in Figure 5.38.

Figure 5.38



The **Plane Slice** group is used to examine two-dimensional plots taken from the **XZ, XY,** and **YZ** plane. The appropriate two-dimensional plot is specified by selecting one of the radio buttons in the **Plane Slice** group. The two-dimensional plot can be examined by pressing the ***View*** button. A sample plot using the plane slice method is shown in **Figure 5.39**.

Figure 5.39



The planar slice can be moved through the plot by pressing the right and left arrow buttons located within the **Plane Slice** group.

5.4 Object Status Window/ Trace Window

*Note that the **Object Status Window** and **Trace Window** cannot be opened separately.*

At the bottom of the HYSYS DeskTop, there is a Window that is, by default, closed. The Window, which is split vertically into two panes, displays status messages and detailed Solver information. The left pane is referred to as the Object Status Window and the right pane is the Trace Window.

5.4.1 Opening and Sizing the Windows

To open the Object Status and Trace Windows, position the mouse pointer on any part of the extra thick border directly above the Status Bar. When the cursor changes to a sizing arrowhead (double-headed arrow), click and drag the border vertically.

If the cursor is placed over the vertical double line which separates the two panes, a horizontal sizing arrowhead appears. The size of the two panes can be adjusted by clicking and horizontally dragging the cursor.

5.4.2 Available Information

The **Object Status Window**:

*For the sake of clarity, a status messages that is yellow in a property view is shown in black in the **Object Status Window**.*

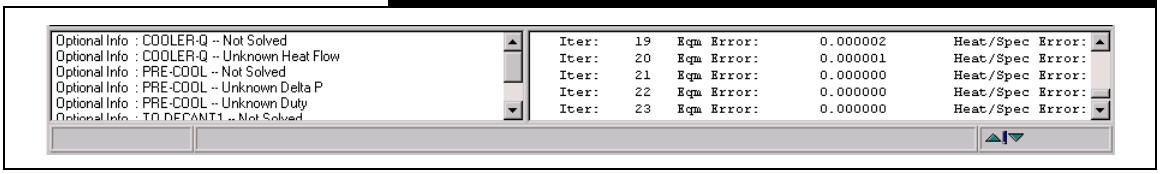
- Shows current status messages for objects in the Flowsheet, coloured accordingly. The colour of the status message for an object matches the colour of the status message on the object's property view.
- Allows you to access the property view of an object described in the status message by double clicking on the message.

The **Trace Window** has three main functions:

- It displays iterative calculations for certain operations, such as the Adjust, Recycle, Reactor, etc. These are shown in black.
- It displays scripting commands, shown in blue.
- If an operation has an error or warning, but still solves, this message is shown in red.

An example of the contents shown in the Object Status and Trace Windows is displayed in Figure 5.40. Each window has a vertical scroll bar, which allows you to move through the contents of the particular window.

Figure 5.40



5.4.3 Object Inspection Options

The options available through the object inspection of the Object Status Window and Trace Window are specific to each pane.

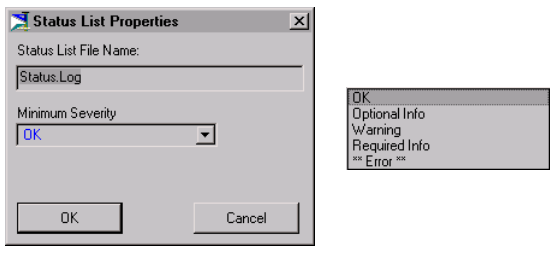
Object Status Window

*Status messages that are **OK** do not appear in the Object Status Window.*

The following options are available by object inspecting the **Object Status Window**:

Option	Description
View Status List Properties	Opens the **Status List Properties** dialog. This view contains an input box for the **Status List File Name** (by default Status.Log), which allows the contents of the left pane to be written to a file. Also on this view is a drop down list for the **Minimum Severity**. From top to bottom, the options in the drop down list represent increasing status message severity. For instance, selecting **Warning** from the list will allow all messages that are warnings or more severe to appear in the left pane. To allow only error messages, which are the most severe, to appear, select the ****Error**** option.
Dump Current Status List to File	Automatically dumps the contents of the left pane to the **Status List File Name**.

Figure 5.41

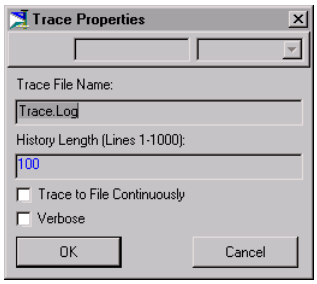


Trace Window

The object inspection options for the Trace Window are:

Option	Description
View Trace Properties	Opens the **Trace Properties** dialog, which contains the following: • **Trace File Name** cell - displays the file name to which the contents of the Trace Window can be written (by default **Trace.Log**). • **History Length** cell - represents the number of lines that the Trace Window keeps in its history. • **Trace to File Continuously** check box - when activated, the Trace Window contents are written to the Trace File. • **Verbose** check box - when activated, the Trace Window displays solver information for all the operations in the case.
Dump Current Trace to File	Automatically dumps the contents of the Trace Window to the Trace File.
Clear Trace Window	Clears all the information from the Trace Window.

Figure 5.42



5.5 Attaching Utilities

The Utilities available in HYSYS are a set of useful tools that interact with your process, providing additional information or analysis of streams or operations. In HYSYS, a utility becomes a permanent part of the Flowsheet, automatically calculating when conditions change in the stream or operation to which it is attached. Access **Utilities** by:

- Selecting Utilities under Tools in the Menu Bar.
- Using the hot key **CTRL U**.
- Choosing the Utilities page in the Attachments tab of a Stream Property View and pressing the ***Create*** button.

There are eight Utilities available from the Stream property view.

A Stream of known composition is required to use these utilities.

*Refer to **Chapter 8 - Utilities** for detailed information concerning the individual utilities.*

Option	Description
CO2 Solids	Determines CO2 freezing conditions.
Cold Properties	Calculates several Cold Properties (e.g. Vapour Pressure, Flash Point, Pour Point, Refractive Index, etc.).
Envelope	Shows critical values and phase envelopes.
Hydrate Formation	Determines hydrate formation conditions.
Property Table	Calculates dependent variables for user specified independent variable ranges or values.
User Property	Allows you to define a property which is dependent on the composition of the stream.
BP Curves	Provides the results of a laboratory style cut point analysis for your simulation streams.
Critical Property	Calculates both true and pseudo critical temperature, pressure, volume and compressibility.

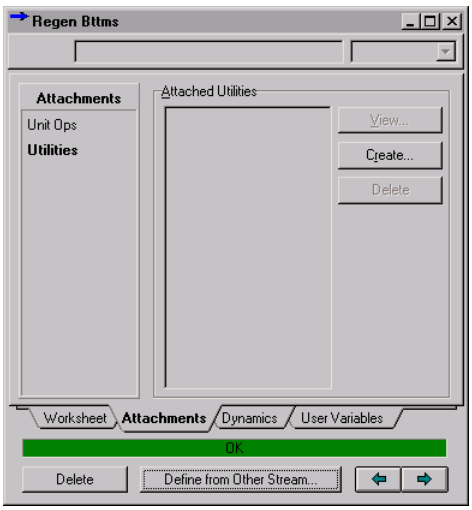
5.5.1 Adding a Utility from a Stream Property View

Not all Utilities can be accessed via the Stream property view, such as Depressuring, Separator Sizing, etc.

To add a Utility from a Stream property view, follow this procedure:

1. Access the property view for the desired stream and select the **Utilities** page of the **Attachments** tab. The **Attached Utilities** group displays any utilities already attached to the stream.

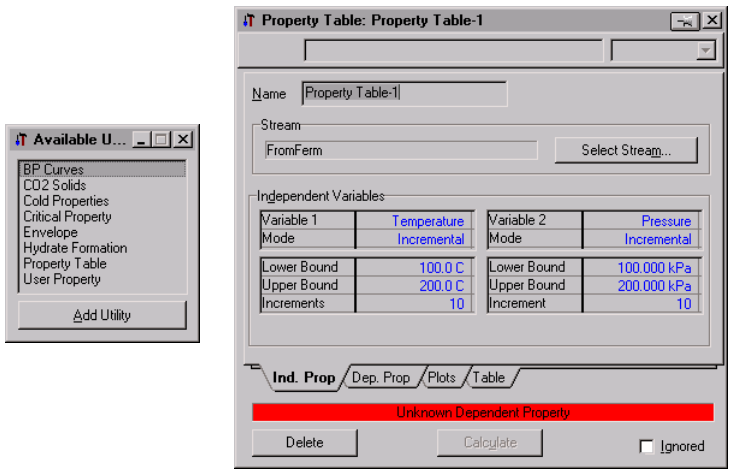
Figure 5.43



2. Press the ***Create*** button to add a new utility.

The **Available Utilities** view appears, from which you can select a utility and press the ***Add Utility*** button. The property view for the selected utility will appear. Note that the **Stream** cell on the Utility view shows the name of the stream from which the utility was created.

Figure 5.44



*For more information on the Utilities available in HYSYS, refer to **Chapter 8 - Utilities**.*

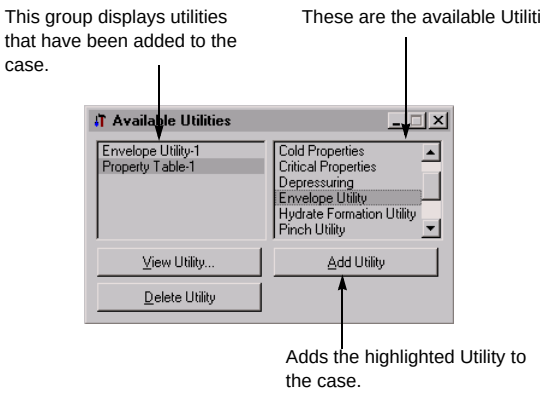
5.5.2 Adding a Utility From the Flowsheet

Once a Utility has been attached to a stream (either through the stream property view or as described in the following example) it remains attached. You can then access the Utility from the stream directly, or through the **Utility** option in the Menu Bar.

To illustrate how to add a Utility, the following is an example of attaching an Envelope to a stream.

1. Open the **Available Utilities** View by selecting **Utilities** under **Tools** in the Menu Bar. (See **Figure 5.45**)

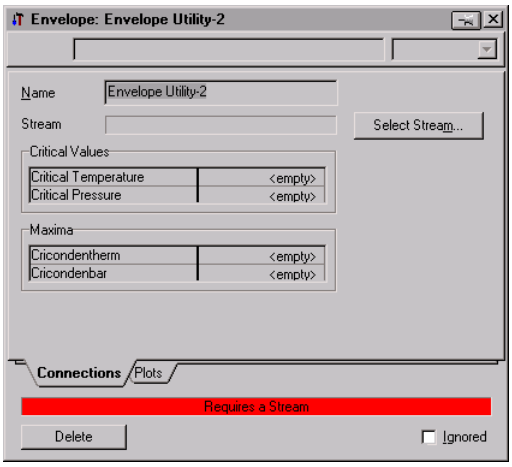
Figure 5.45



2. Select the Utility you want to add from the **Available Utilities** view (right hand side). In the above view, the **Envelope** is selected as the utility to add.
3. Press the ***Add Utility*** button. HYSYS displays the view for the Utility.
4. If desired, change the name of the utility in the **Name** cell.
5. Press the ***Select Stream*** button to access the Object Navigator.

*Refer to Section 5.2.1 - Object Navigator, **Selecting an Object for a Utility** for information on selecting a stream.*

Figure 5.46

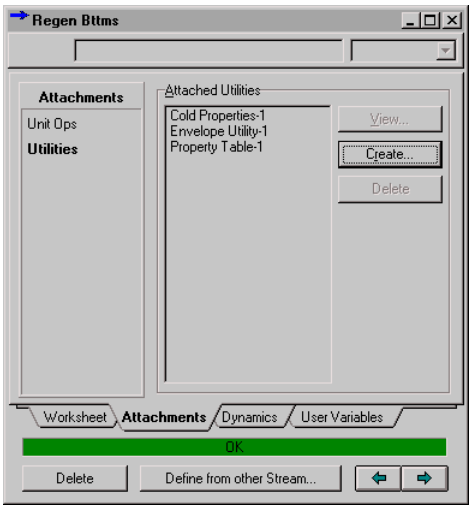
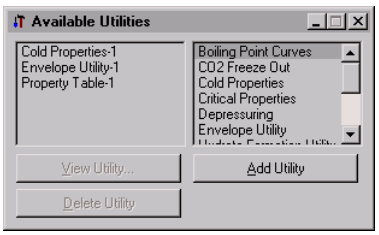


6. Select a stream from the Object Navigator and press ***OK***.

7. As soon as the stream is attached, HYSYS moves to the next page of the Envelope view, the **Plots** page and displays the Pressure - Temperature phase envelope.
8. Press the ***Close*** button to exit this view.

5.5.3 Viewing Utilities

Note that the property view for a Utility can remain open independently of the stream to which it is attached. There are a three ways within HYSYS to View Utilities.

Method	Description	View
Stream Property View	Access the Stream property view for the stream to which the utility is attached. Open the **Utilities** page in the **Attachments** tab of the Stream property view. HYSYS displays any utilities attached to the stream in the **Attached Utilities** group. Notice that a stream can be attached to more than one Utility. Select the Utility you want to view and press the ***View*** button.	
Available Utilities View	Access this view by selecting **Utilities** under **Tools** in the Menu Bar, or through the hot key **CTRL U.** From the view, select a Utility (from the left side) and press the ***View Utility*** button.	
Workbook Page	Any utility attached to the highlighted stream is shown in the box in the bottom left hand corner of the Workbook Page. Double click on the Utility name in this box to display the Utility property view.	

5.5.4 Deleting Utilities

Once you delete a Utility, it cannot be recovered.

The procedures for deleting and viewing utilities are very similar. To delete a utility, use one of the following methods:

Method	Description
Available Utilities View	When you are in the **Available Utilities** view, select the Utility you want to delete. Press the ***Delete Utility*** button to delete it from the case.
Stream Property View	Open the **Utilities** page in the **Attachments** tab of the Stream property view. Select the Utility in the **Attached Utilities** group and press the ***Delete*** button.
Utility Property View	Open the property view for the specific utility (e.g., CO2 Solids, Cold Properties, etc.). Press the ***Delete*** button to delete the Utility.

5.5.5 Attaching a Utility to a Different Stream

Once a Utility is created, you can change the stream to which it is connected on the **Connections** page of the Utility property view. For example, on the **Connections** page of the **Envelope** utility (shown previously), press the ***Select Stream*** button. Change the stream to which the utility is attached in the Object Navigator view and press ***OK***. Once you change a connection, the name of the utility will automatically appear on the **Utilities** page of the new stream's property view.

6 Output Control

6.1 Printing in HYSYS 4

6.1.1 Menu Bar Options 4
6.1.2 Printing Specsheets 5
6.1.3 Printing the PFD 7
6.1.4 Printing Plots 8
6.1.5 Printer Setup 9

6.2 Reports 10

6.2.1 Creating a Report 11
6.2.2 Editing a Report 14
6.2.3 Deleting a Report 15
6.2.4 Printing and Previewing Reports 15

6.3 Printing the PFD as a DXF File 16

6.4 Graph Control 18

6.4.1 Data Page Tab 19
6.4.2 Axes Tab 20
6.4.3 Title Tab 21
6.4.4 Legend Tab 22
6.4.5 Plot Area Tab 23

6.5 Format Editor 23

In HYSYS, the capability to effectively transcribe process information concerning your simulation case is readily available through printing features. By allowing you to obtain information in various levels of detail, you can efficiently create printed reports that can range from basic data to comprehensive summaries. There are two primary printing options in HYSYS:

- **Object Specific**
- **Reports**

Object Specific printing relates to the object that currently has focus in the simulation case. For instance, if the active location resides on a **SEPARATOR** property view, you can print a Snapshot of the property view as it is seen on your monitor or print out a Specsheet specific to the unit operation. A Specsheet displays object related information, which can include input specifications and calculated results. Each object within HYSYS has at least one Specsheet available, with many objects also having condensed versions of the full Specsheet.

The printing of Reports deals with much more extensive sets of information. When creating a Report, you collect the Specsheets of multiple objects in one document. HYSYS allows you to select any Specsheet for any object currently in your simulation case. For each Report that you create, you have the option of customizing the Page Setup.

Both primary printing features can be accessed via the Menu Bar. However, Object Specific printing is also available via Object Inspection. By Object Inspecting the Title Bar of an object, you can print or preview its associated Specsheet.

The high degree of flexibility offered by the design of HYSYS is once again demonstrated with its printing features. Both Specsheet and Report views can remain open while you manipulate your Flowsheet. If changes occur which affect the values shown in a Specsheet, you can easily update the information.

This chapter provides more detail on all of the printing features available in HYSYS.

6.1 Printing in HYSYS

*Refer to **Section 6.1.1 - Menu Bar Options** for details.*

*Refer to **Section 6.2 - Reports** for details on the Report Manager.*

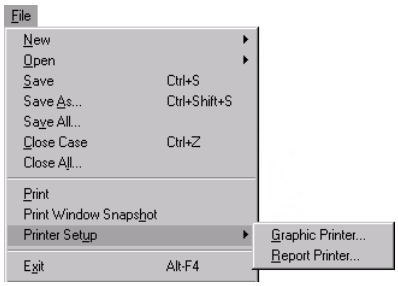
In HYSYS, you can generate printed information in one of three ways:

Method	Description
Menu Bar	Select one of the options under **File** in the Menu Bar.
Object Inspection	Object Inspect the Title Bar of a view and select the **Print Specsheet** option from the menu. A list of available Specsheets is displayed for the object. Object inspection of the Title Bar is the same as selecting **Print** under **File** in the Menu Bar. However, object inspection proves useful when views are Modal, since the Menu Bar cannot be accessed.
Report Manager	Select **Reports** under **Tools** in the Menu Bar to access the **Report Manager** view.

6.1.1 Menu Bar Options

The HYSYS Printing options which can be accessed under **File** in the Menu Bar are:

Figure 6.1



Option	Description
Print	Lists the available Specsheet(s) for the currently active object. You can highlight a Specsheet and either preview or print it. See **Section 6.1.2 - Printing Specsheets** for more detail.
Print Snapshot	Prints a bitmap of the currently active HYSYS view. Use this option when you want to print a view that does not have a Specsheet associated with it (i.e. the Optimizer view or a Table such as a Column Profiles Table).
Printer Setup	With this option, you can select either the **Graphic Printer** or the **Report Printer**. This allows you to select the printer, paper orientation, paper size and source. Refer to **Section 6.2.4 - Printing and Previewing Reports** for details.

6.1.2 Printing Specsheets

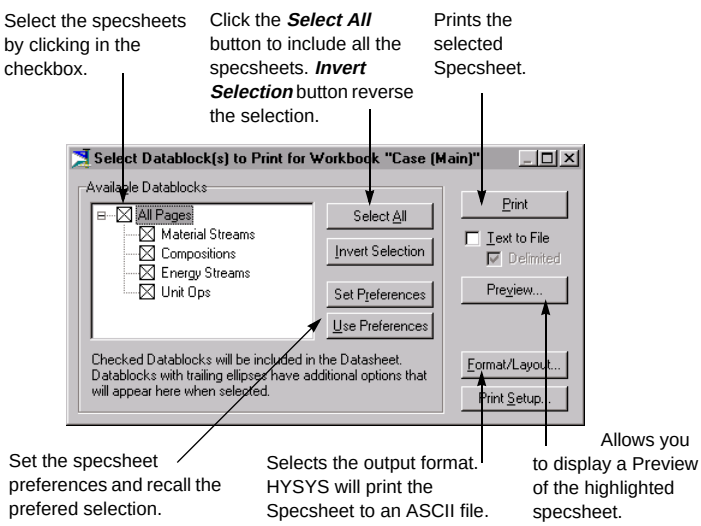
All default and customized Workbook pages have Specsheets available.

When you choose the **Print** option, HYSYS automatically prepares to print the Specsheet for the view that currently has focus. For instance, this could be a Workbook tab, a Stream, a Unit Operation or a Utility. If you are currently in an area where printing is not allowed (e.g. the Optimizer), you will be informed that no Specsheets are available for the selected object when you choose **Print**.

Printing Workbook Specsheets

When a Workbook tab is the active location and you select the **Print** option, the following view appears.

Figure 6.2

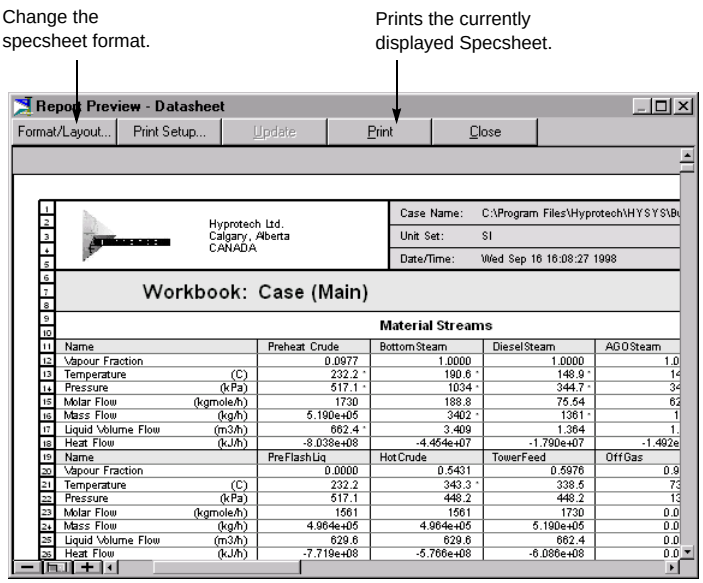


The active location in the Flowsheet governs which Specsheets will be available. In this case, the active location is a Workbook tab, thus the only available options pertain to the Workbook. The **All Pages** Specsheet, displays all the information in the Workbook. Only the variables present on the Workbook tab appear in the Specsheet. If you want additional variables to appear on the Specsheet, you have to add variables to the Workbook tab.

*See **Section 4.5.2 - Editing a Workbook Tab** for details on adding variables to a Workbook page.*

*The **Zoom** buttons in the Preview have the same function as in the PFD.*

Figure 6.3



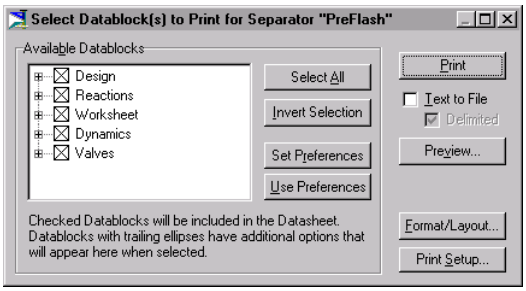
Printing Operation Specsheets

When you want to print the Specsheet for an operation, you must make that operation the active location, i.e. open the operation Property View. When you select the **Print** option under **File** in the Menu Bar, the **Select Datablock(s) to Print for** view appears.

*You may encounter particular Specsheets for certain operations. For instance, a column tray section has a **Tray Section Component Summary**, and a Strip Chart has a **Strip Chart Variables and Historical Data** Specsheet.*

This view is similar to the view shown for the Workbook. The only difference is the specsheets listed in the **Available Datablock** group.

Figure 6.4

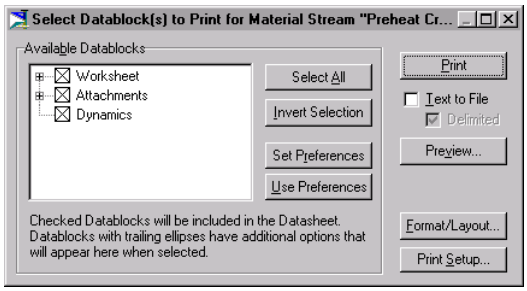


Printing Stream Specsheets

When you select **Print** under **File** in the Menu Bar while a material or energy stream has focus, the **Select Datablock(s) to Print for** view will be shown.

The same options available for the operation Specsheets are listed as stream Specsheets and the only difference in the specsheets listed in the **Available Datablock** group.

Figure 6.5

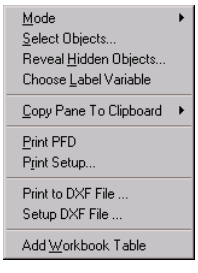


6.1.3 Printing the PFD

HYSYS also allows you to print the PFD. To print the PFD, use object inspection (secondary mouse button on an empty area of the PFD). This will produce the menu shown in Figure 6.6, which has three print related functions:

Alternatively, you may select these printing options under ***File*** *in the Menu Bar.*

Figure 6.6



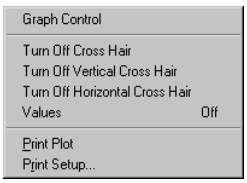
Option	Description
Print PFD	Prints the PFD as it appears on the screen. Only the sections visible within the PFD window will be printed. Any Tables that you have added will also be printed. Note that when you select this option, the PFD is immediately printed without accessing any further menus.

Option	Description
Print Setup	Accesses the typical Windows Print Setup, which allows you to select the printer, the paper orientation, the paper size and paper source.
Print to DXF	Prints the entire PFD to a dxf file. For more information refer to **Section 6.3 - Printing the PFD as a DXF File**.

6.1.4 Printing Plots

To print a plot, object inspect the plot area and select one of the two options for printing:

Figure 6.7



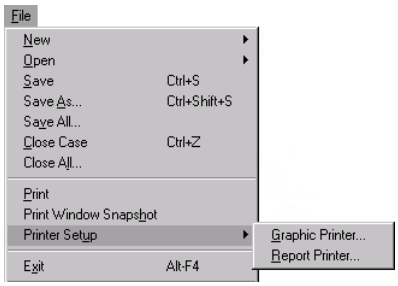
*For information on controlling the appearance of plots, refer to **Section 6.4 - Graph Control**.*

Object	Description
Print Plot	Prints the plot as it appears on the screen.
Print Setup	Accesses the typical Windows Print Setup, which allows you to select the printer, the paper orientation, the paper size and paper source.

6.1.5 Printer Setup

The **Print Setup** view can be accessed by selecting **Printer Setup** under **File** in the Menu Bar. The side menu then provides two options:

Figure 6.8

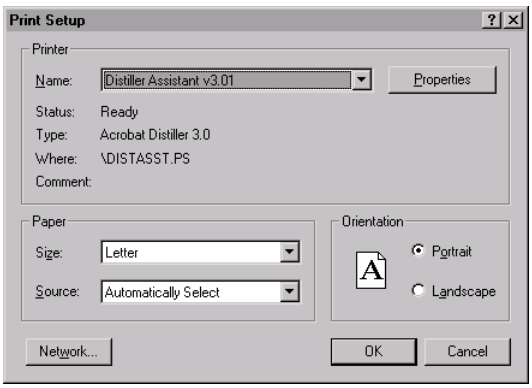


Printer	Description
Graphic Printer	Use the **Graphic Printer** when printing the PFD, Plots, Strip Charts and Snapshots.
Report Printer	The **Report Printer** is used for Specsheets, Reports and Text.

If the **Print Setup** view is accessed through object inspection (i.e. via the PFD), HYSYS will default to the appropriate printer according to the active location in the Flowsheet.

On the **Print Setup** view, you can select the printer, as well as the page orientation, paper size and paper source. The layout of the **Print Setup** view can vary depending on the selected printer. You can also modify the default properties for the selected printer by pressing the ***Properties*** button.

Figure 6.9

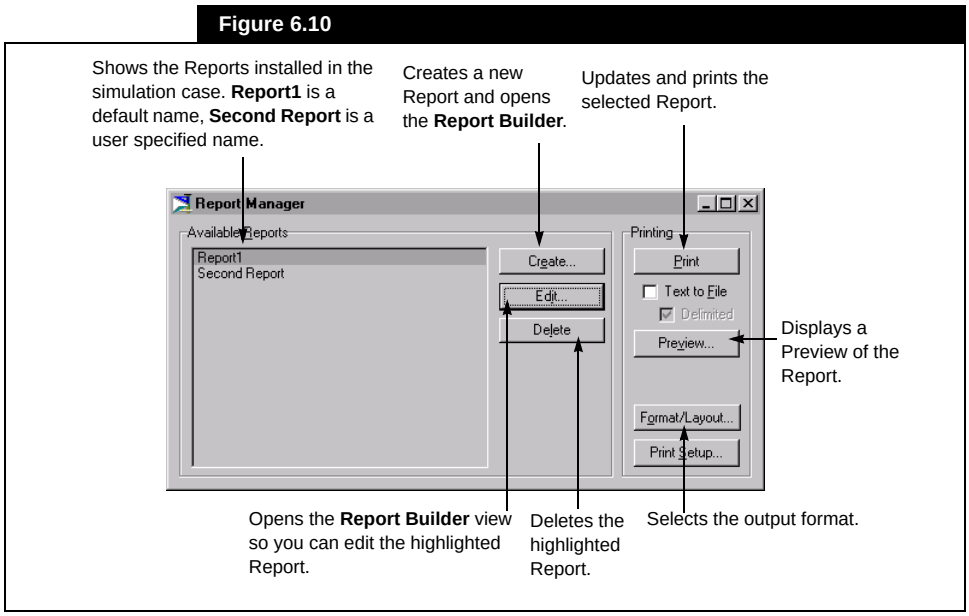


6.2 Reports

Within a simulation case, you can print stream and operation Specsheets with the **Print** function. However, this only allows you to print Specsheets for a single object at a time. By using the **Report Manager**, you can add multiple Specsheets for streams and operations to a single Report and print the entire document at once. You can also format the display of the Report to meet your personal preferences.

The **Report Manager** is accessed by selecting **Reports** under **Tools** in the Menu Bar or with the hot key combination CTRL R. All reports that you have created for the simulation will be listed in the **Reports** group.

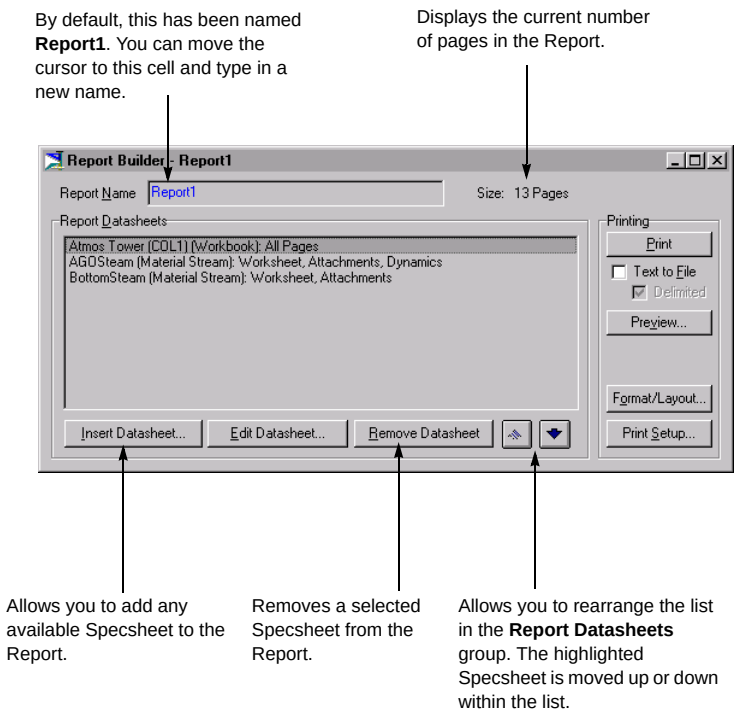
Figure 6.10



6.2.1 Creating a Report

By pressing the ***Create*** button in the **Report Manager**, the **Report Builder** view will be displayed for a new Report. The four buttons in the **Printing** group, ***Print, Preview, Format/Layout*** and ***Print Setup*** provide the same functions as those in the **Report Manager**.

Figure 6.11



Contents Tab

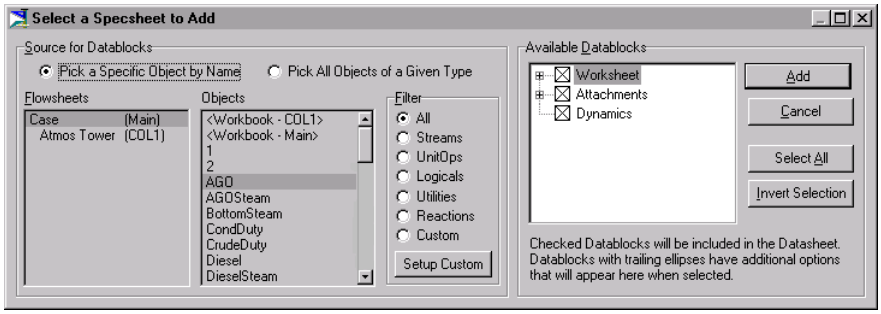
Inserting a Specsheet

When you press the ***Insert Datasheet*** button on the **Report Builder** view, the **Select a Specsheet to Add** view appears (see Figure 6.12).

The view can be analyzed by examining its two main groups. The first is the **Source for Specsheets** group, which contains two radio buttons. Depending on which radio button is selected, the information contained within the group will change. In the following sections, the functionality of each of the radio buttons is explained.

The second group is the **Available Specsheets** group, which lists the Specsheets that can be added to the Report. The Specsheet options that are displayed in this group depend upon the selection made in the **Source for Specsheets** group.

Figure 6.12



Pick a Specific Object by Name

When this radio button is selected, the **Select a Specsheet to Add** view, as shown above, is displayed. You can insert individual Specsheets for any object that is present in the simulation case.

*The **Filter** can be used to reduce the list of **Flowsheet Objects**. The **Filter** group is only present when the **Pick a Specific Object by Name** radio button is selected.*

Use the following procedure:

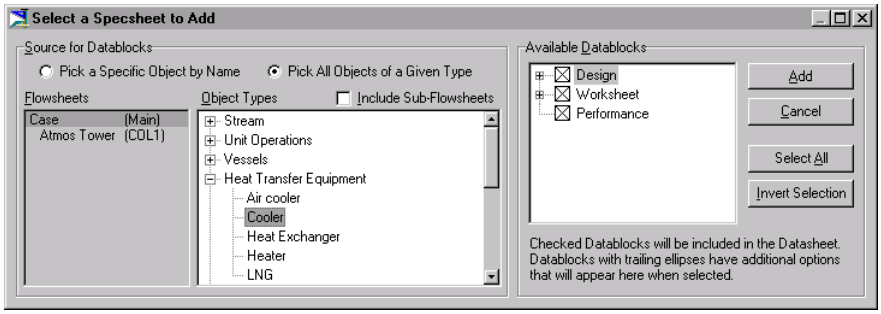
1. From the **Flowsheets** group, pick the Flowsheet that contains the object.
2. Choose the desired object (stream, operation or logical) from the **Objects** group. In the above view, **AGO** is selected.
3. In the **Available Datablocks** group, all Specsheets available for the selected Flowsheet Object are displayed. Select the Specsheet that you want to insert in the Report.
4. Press the ***Add*** button.
5. Repeat steps #1 through #4 for each Specsheet that you want to add to the Report.
6. When all Specsheets have been added, press the ***Done*** button.

*To return to the **Report Builder** view without adding a Specsheet, press the **Cancel** button.*

Pick All Objects of a Given Type

When this radio button is selected, you can choose a specific object type in the **Source for Specsheets** group and then pick a Specsheet in the **Available Specsheets** group. HYSYS adds a set of Specsheets to the Report; one Specsheet is added for each occurrence of the specific Object Type in the case.

Figure 6.13



Follow this procedure to add a group of Specsheets:

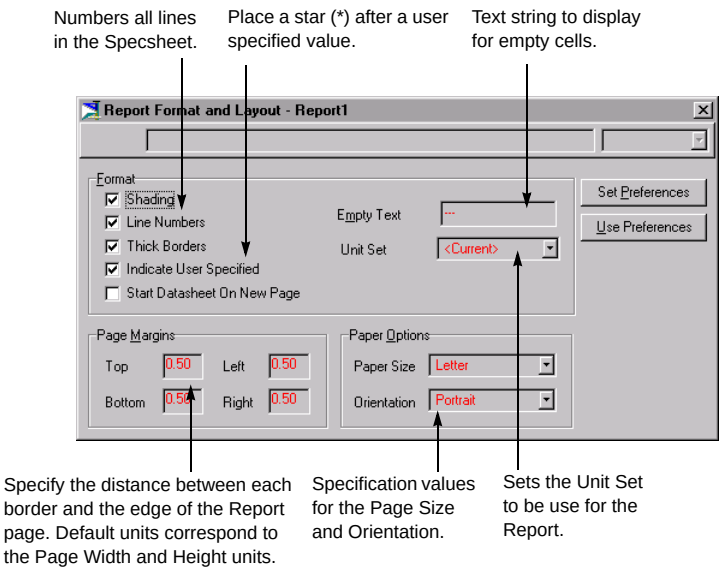
1. In the **Object Types** group, select a Stream or Unit Operation type. The '+' symbol indicates that there are additional sub-items underneath. To open the list of sub-items, double click on the object, or click on the '+'. Once the list is open, the '+' symbol will be replaced with '-'. To close the list, double click on the main object or click on the '-' symbol. If you select the object types **Streams** or **Unit Operations**, a specsheet for every stream or unit operation in the selected Flowsheet will be added to the report.
2. For the selected Object Type, you can add Specsheets to the Report for objects that reside within Sub-Flowsheets. Simply activate the **Include Sub-Flowsheets** check box.
3. Select a Specsheet in the **Available Datablocks** group.
4. Press the ***Add*** button.
5. Repeat steps #1 through #4 for each group of Specsheets that you wish to add to the Report.
6. Press the ***Done*** button when no more Specsheets are to be added.

Page Setup Tab

The default Report format is set in the Session Preferences. See Section 7.7.5 - Preferences for more detail.

On the **Report Builder** view, selecting the **Page Setup** tab will display the options available for customizing the format of the Report.

Figure 6.14



6.2.2 Editing a Report

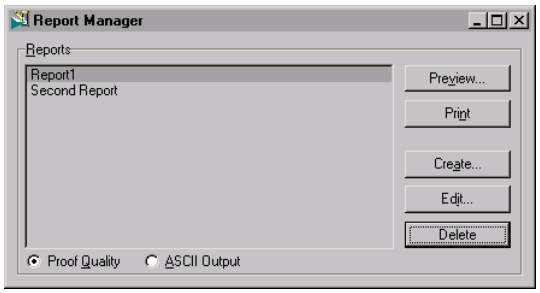
When a new Report is created, all the necessary Specsheets can be added and the custom formats set. However, if you close the **Report Builder** view and then want to change some of the input in the Report, you can do so using the Editing feature.

Highlight a Report on the **Report Manager** view and press the ***Edit*** button. The **Report Builder** view opens, from which you can edit the list of Specsheets or the format of the Report.

6.2.3 Deleting a Report

You can delete a report from either the **Report Manager** view or the **Report Builder** view. When in the **Report Manager** view, highlight the report to be deleted and press the ***Delete*** button.

Figure 6.15



If the **Report Builder** view is active, pressing the ***Delete*** button will delete the current report. No confirmation is required for deleting reports, so ensure that the correct report is selected.

6.2.4 Printing and Previewing Reports

A Report can be Printed or Previewed from either the **Report Manager** view or the **Report Builder** view. If the **Report Builder** view is open, press the ***Print*** button and the entire Report is updated and printed. When you **Preview** a Report, a ***Print*** button is always available on the Preview screen, so you can print directly from the preview. From the **Report Manager** view, you are given the option to **Print** to a printer or transfer the Report contents to an ASCII file.

6.3 Printing the PFD as a DXF File

*To open the .dxf file in AutoCAD Release 14, select **DXF** in the **Files of Type** drop down on the **File Open** window.*

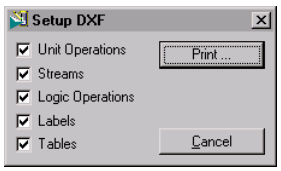
An ASCII Drawing Interchange File (DXF) representing the HYSYS PFD can be created by HYSYS. The DXF can then be read into AutoCAD. By default, the file created is called *pfd.dxf* and will contain the entire PFD regardless of what is visible on the screen. A different layer is generated for each of the following groups:

- **Physical unit operations** (pfdOP)
- **Logical unit operations** (pfdLOGICOP)
- **Streams** (pfdSTREAM)
- **Stream labels** (pfdLABEL)
- **Table or other text** (pfdTABLE)

Each layer is created using standard AUTOCAD blocks.

To create a DXF file in HYSYS, object inspect the PFD and select **Print to DXF File** from the menu. A standard windows file selection dialog named **Write DXF File** will appear, from which you can select the path and file name for the DXF file.

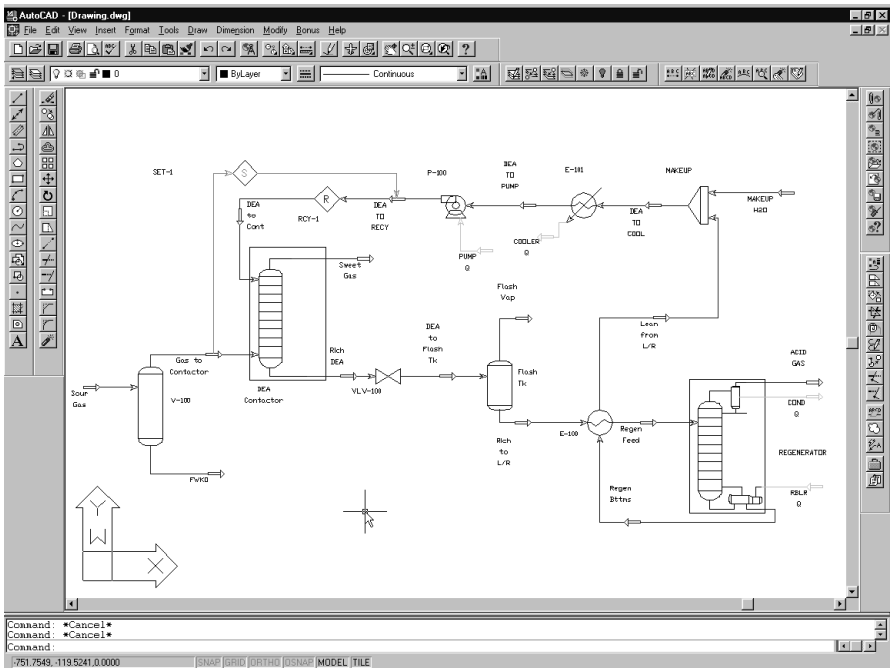
Figure 6.16



You can also object inspect the PFD and select **Setup DXF File**, which allows you to set which layers are sent to the file. Activate the corresponding check box for each layer that you want in the file.

An example of a *pfd.DXF* exported into AutoCAD is shown below:

Figure 6.17



Included in the directory **\HYSYS\Support** is a header file named *autocad.hdr*. If you have trouble transferring the *.dxf* into AutoCAD, changes may be needed to this header file. The default version of the header file may not be compatible with certain versions of AutoCAD and therefore, may need to be manipulated.

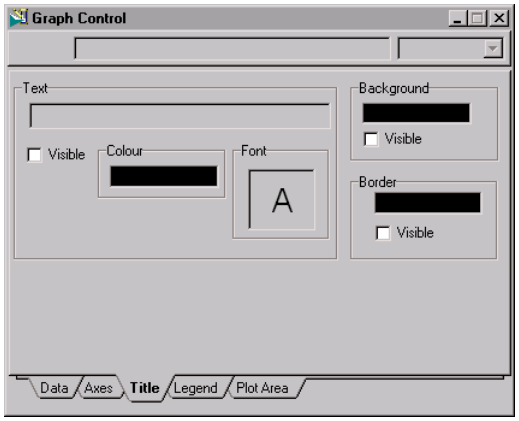
Try the following procedure to check the header file:

1. Start a completely new AutoCAD drawing.
2. Note that the custom corporate title blocks and border should not be in the blank drawing that is used to generate the test header.
3. Save the blank drawing as a DXF file using AutoCAD's DXFOUT command.
4. Compare the DXF file of the blank drawing to the AutoCAD.hdr supplied with HYSYS.

6.4 Graph Control

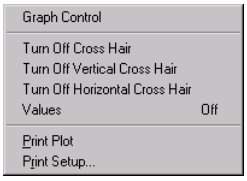
You can customize each individual plot in HYSYS using the Graph Control tool. You can modify many of the plot characteristics, which are categorized into the five tabs of the Graph Control property view: **Data**, **Axes**, **Title**, **Legend** and **Plot Area**.

Figure 6.18



There is more than one way to access the Graph Control property view. The easiest method is to object inspect any spot on an active plot and select **Graph Control** from the object inspection menu.

Figure 6.19



You can also open the Graph Control property view to a specific tab. By double clicking the primary mouse button in the plot area, you make the plot the active view. Then, either double click on the plot **Title** or on the **Legend** to access the respective tab of the Graph Control tool.

While the plot area has focus, you can also click and drag on either the **Legend** or **Title** to reposition the selected item.

The changes made to a plot using the Graph Control tool are specific to the active plot. To make global changes, which will affect all plots, use the **Colours** and **Fonts** pages on the **Resources** tab of the Session Preferences property view.

6.4.1 Data Page Tab

For each data set on the plot, you can:

- View the data set Type.
- Modify the data set Name.
- Specify the Colour and Symbol that represent the data on the plot.
- Choose a Line Style.
- Show/hide the Symbol or Line.
- Show/hide the name in the Legend.

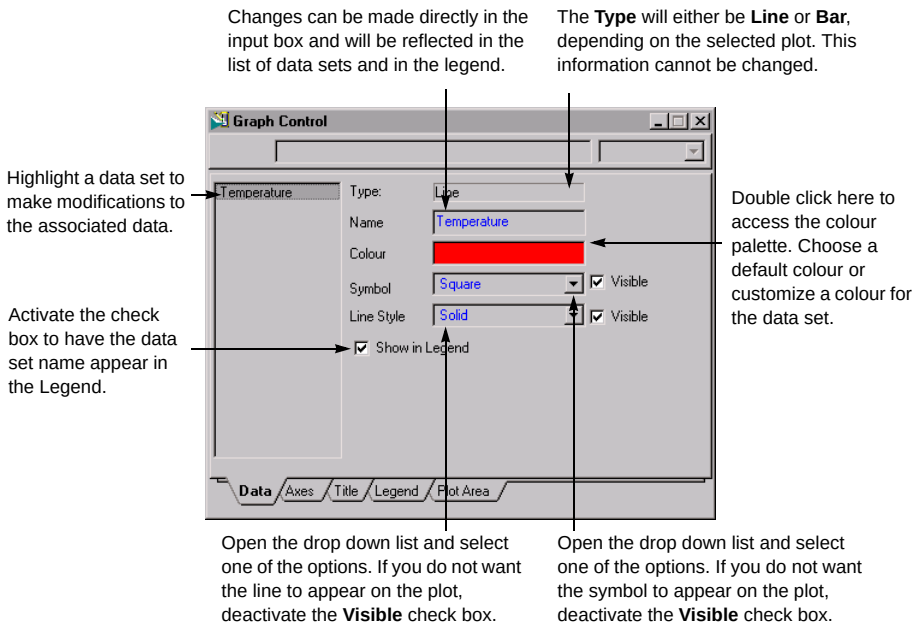
All changes instantaneously affect the plot. There is no need to close the Graph Control property view to view the modifications.

Any changes that are made will affect only the data set that is highlighted in the list.

*The Symbol drop down is not available for **Bar** Charts.*

To make modifications to the appearance of a data set, highlight the name of the set in the list of data sets. The information which corresponds to the highlighted data set will appear, as shown in Figure 6.20.

Figure 6.20



6.4.2 Axes Tab

From the **Axes** tab, you can do the following for each axis:

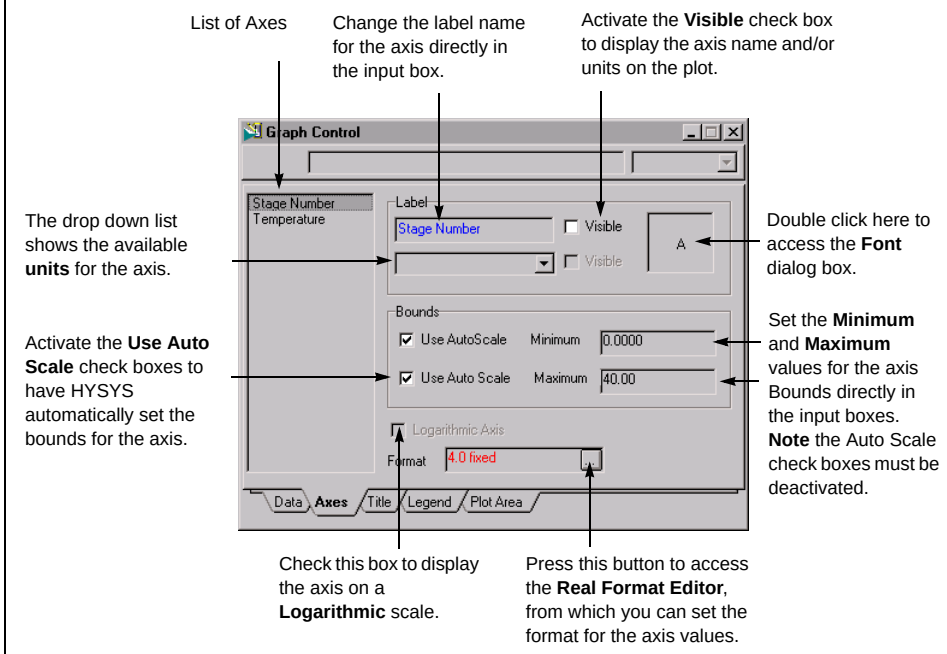
- Make changes to the Label name, font and units.
- Show/Hide the Label name and/or Units.
- Define the axis Bounds or use the Auto Scale function.
- Format the axis values.

Refer to Section 6.5 - Format Editor for information regarding value formatting.

Any changes that are made will affect only the axis that is highlighted in the list.

To make modifications to the appearance of an axis, highlight the name in the list of axes. The information that corresponds to the highlighted axis will appear.

Figure 6.21

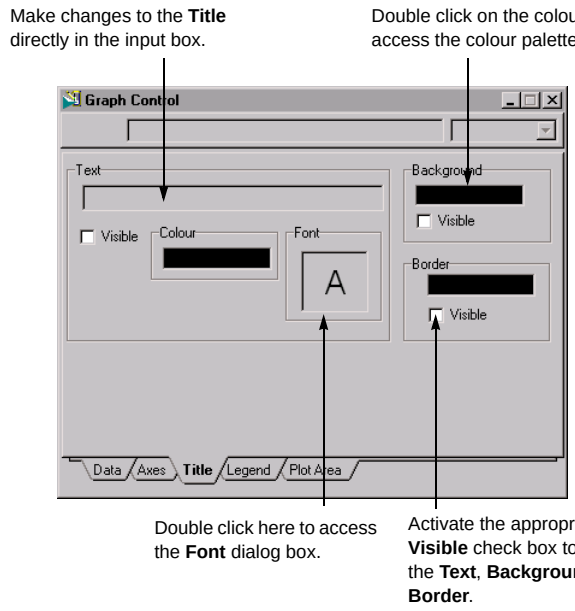


6.4.3 Title Tab

On the **Title** tab, you can:

- Change the default title name and font.
- Select the colours for the Text, Background and Border of the title.
- Show/Hide the Title, Background and Border.

Figure 6.22

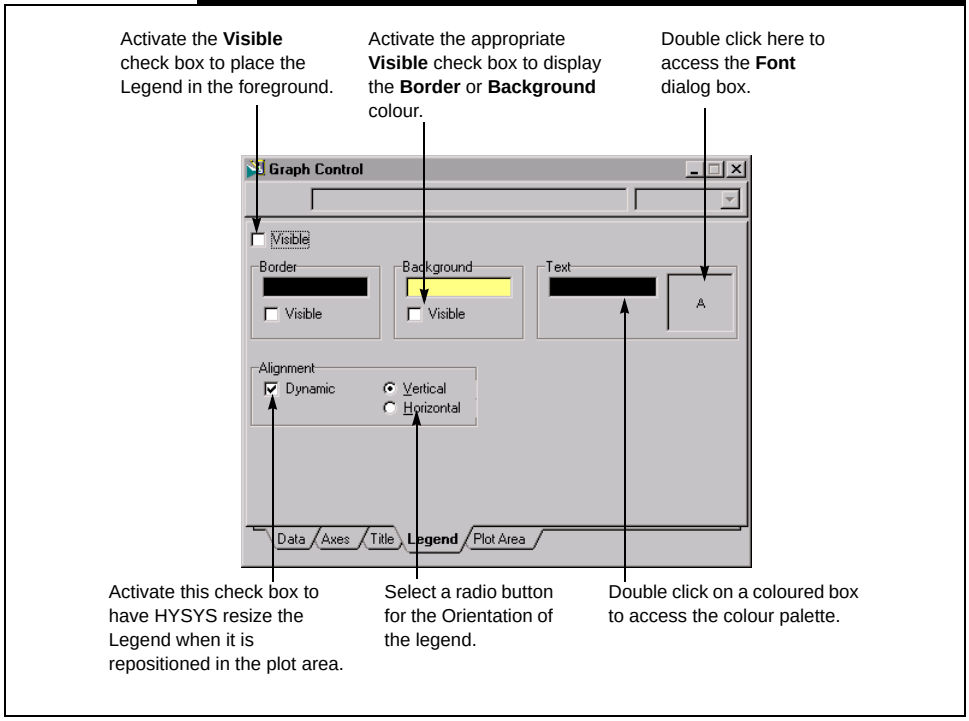


6.4.4 Legend Tab

The **Legend** tab allows you to change the appearance and location of the legend. You may:

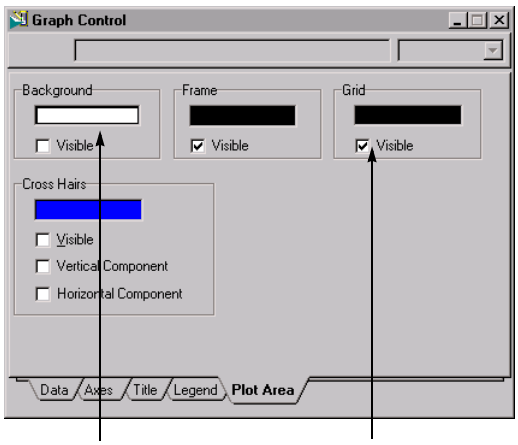
- Change the colour of the Border, Background or Text.
- Choose the orientation: Vertical or Horizontal.
- Show/Hide the Border and Background.
- Enable automatic resizing of the legend upon repositioning.
- Place the legend in the foreground or background.

Figure 6.23



6.4.5 Plot Area Tab

Figure 6.24



From the **Plot Area** tab, you can:

- Change the colour of the Background, Frame or Grid.
- Show/Hide the Background, Frame or Grid.

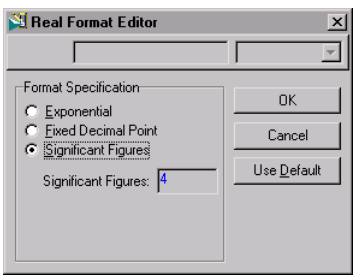
6.5 Format Editor

The Real Format Editor can be accessed from the following views:

- Workbook Setup property view
- Graph Control property view

*The layout of the **Real Format Editor** view is dependent upon the radio button selection.*

Figure 6.25

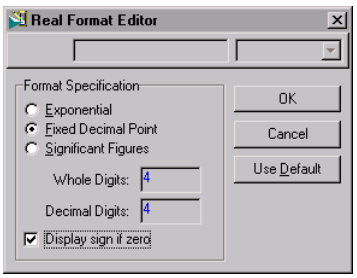


From the view of Figure 6.25, you can set the format of values that are displayed in HYSYS. The **Format Specification** group uses three radio buttons to list the options available:

Format	Description
Exponential	The values will be in exponential form with a specified number of **Significant** digits. For instance 8546 would be displayed as 8.546e+03 if 4 was specified in the **Significant** cell.
Fixed Decimal Point	Specify the maximum number of digits that will be displayed before the decimal point in the **Whole** cell - see Figure 6.26. If the **Whole** cell limit is exceeded by a value in HYSYS, exponential form will be used. In the **Decimal Digits** cell, input the number of digits that will appear after the decimal point. By activating the **Display sign if zero** check box, HYSYS will show a '**+**' symbol in front of a value that is displayed as zero using the current precision. For instance, a composition of 0.000008 would be shown as zero when using a **Decimal Digits** value of 4. With the check box activated, the cell would show **+0.0000** to signify that there is a small number present.
Significant Figures	In the **Significant** cell, specify the number of significant figures (between 0 and 9) that you wish to display.

The ***Use Default*** button closes the **Real Format Editor** view and assigns the HYSYS default format to the associated values.

Figure 6.26



7 Menu Bar Options

7.1 File 4
7.1.1 Starting a Simulation 4
7.1.2 Saving a Simulation 7
7.1.3 Closing a Simulation 8
7.1.4 Printing 8

7.2 Edit 9

7.3 Simulation 10
7.3.1 Main Properties 11
7.3.2 Optimizer 13
7.3.3 Event Scheduler 14
7.3.4 Integrator 14
7.3.5 View Equations 15
7.3.6 Enter Basis Environment 15

7.4 Flowsheet 17
7.4.1 Reaction Package 18
7.4.2 Fluid Packages 21
7.3.7 Oil Output Settings 16
7.4.3 User Properties 21

7.5 Workbook 25

7.6 PFD 26

7.7 Tools 27
7.7.1 Face Plates 28
7.7.2 DCS 29
7.7.3 Script Manager 30
7.7.4 Macro Language Editor 32
7.7.5 Preferences 32

7.8 Column 59

7.9 Basis 59

7.10 Window 60

7.11 Help 62

Most of the tasks described to this point (i.e. installing streams and operations, accessing the Workbook and PFD, Printing, etc.) have hot keys or buttons associated with them, which provide quick access to their capabilities. Each of these functions can also be accessed through the Menu Bar. The list of command or function groups, which is displayed at the top of the HYSYS DeskTop, operates as a pull down menu system. By selecting one of the options in the Menu Bar, a menu of associated commands or options is opened.

Session Preferences can be accessed through the Menu Bar only.

In addition to the functions already described, the Menu Bar also provides access to a number of functions that can only be accessed through this route. Included in the functions that can only be accessed via the Menu Bar are setting Session Preferences (units, default naming schemes, etc.) and switching to another simulation currently in memory.

In this chapter a description of each of the options that are available through the Menu Bar will be provided. In cases where the options have been described elsewhere in greater detail, only a quick review will be provided. Others, such as Session Preferences and Scripting will be described completely.

You can access the Menu Bar options in three ways:

- Select the desired Menu Bar item using the primary mouse button, which will automatically open the associated menu.
- Use the **ALT** key in combination with the underlined letter in the Menu Bar title. For example, **ALT T** will open the **Tools** menu.
- Use the **ALT** key by itself to move the active location to the **File** option in the Menu Bar. Once the Menu Bar becomes the Active location in HYSYS, you can manoeuvre through it using the keyboard. The up and down arrows move through the menu associated with a specific item, while the left and right arrows move you to the next Menu Bar item, automatically opening the associated menu.

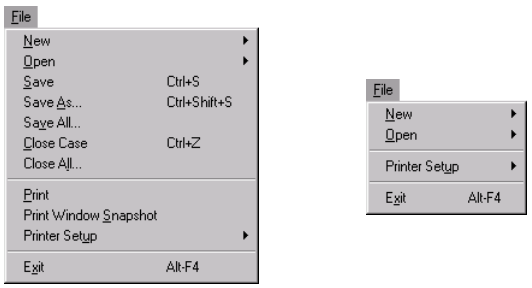
If you want to switch focus from the Menu Bar without making a selection, press the ESC key or the ALT key.

7.1 File

Note that a menu item with an arrow head pointing to the side has further options on a submenu.

There are two variations to the drop down menu under **File**. A condensed menu will be displayed in HYSYS before a simulation has been created or opened. The options which are common to both versions of the menu, as well as the functions specific to the detailed menu will be explained in this section.

Figure 7.1



The menu options can be grouped into four main categories:

- Starting a Simulation
- Saving a Simulation
- Closing a Simulation
- Printing

7.1.1 Starting a Simulation

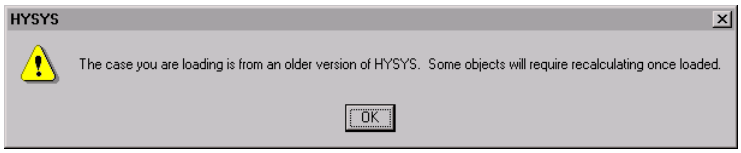
The **New** and **Open** options under **File** in the Menu Bar allow you to create a new Flowsheet or open an existing Flowsheet. If you select either **New** or **Open**, an expandible menu is produced containing the available Flowsheet options:

See Chapter 2 - Flowsheet Architecture for more information on SubFlowsheets and Templates.

Option	Description
Case	Creates a new simulation case or opens an existing one. This option allows you to access HYSYS simulation cases (*.hsc), HYSIM simulation cases (*.sim) or Backup simulation cases (*.bk?).
Template	Creates a new template or opens an existing one. These are Sub-Flowsheet templates.
Column	Creates a new column Flowsheet or opens an existing one.

When opening a case from an older version of HYSYS you will receive the following message.

Figure 7.2

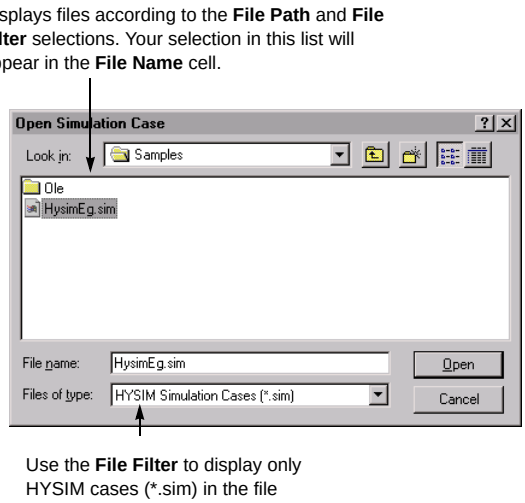


When you receive this message re-enter the Simulation Basis and make sure that all of your assays or reactions are re-calculated.

Reading a HYSIM Case

HYSYS presents the functionality to open HYSIM simulations and to transfer all compatible data into the appropriate HYSYS Environments.

Figure 7.3



*Note that the description for each HYSIM case has a Revision number. If the value shown is **less than 10014**, the HYSIM case **IS NOT VALID** for transfer into HYSYS.*

Open Case Button

To open a HYSIM case, do the following:

1. Under **File** in the Menu Bar, select **Open** and then **Case** from the submenu, or press the ***Open Case*** button.
2. On the **Open Simulation Case** dialog, select **Hysim Simulation Cases (*.sim)** from the **File Filter** drop down list.

3. Choose a directory that contains a HYSIM case in the **File Path** group.
4. Select a HYSIM case in the list of cases or type the name of a case in the **File Name** input cell.
5. Press the ***OK*** button.

HYSIM Functionality NOT Supported In HYSYS

HYSYS does not currently support some HYSIM functionality, therefore it is not possible to transfer all HYSIM information into HYSYS. The following table lists some issues of which you should be aware:

Object	Details Not Supported
Calculator	All Programs
Column	Condenser or Reboiler with Side Stripper Draw
Column	Condenser or Reboiler with Pump Around Draw
Column	Reboiler Liquid Draw (other than Bottoms product)
Column	Condenser Side Vapour Draw
Column	Reboiler Water Draw
Column	In AMSIM, tray efficiencies require the input of tray dimensions on a per tray basis. HYSYS supports only one diameter, one weir length and one weir height per Tray Section. In this case, dimensions of the 2nd stage from the bottom of the HYSIM column are used for the HYSYS Tray Section.
Liquid Liquid Extractor	Pump Arounds
Cyclone	Liquid Streams
Hydrocyclone	Vapour Streams
Rotary Vacuum Filter	Only Connections are transferred. Other operation parameters must be specified in HYSYS.
Baghouse Filter	Liquid Streams
Crystallizer Solid Operation	All
Tee	Energy Stream Attachments
LNG	If the LNG Duty Stream is attached to another operation in HYSIM, the Flowsheet will not be complete in HYSYS.
Plug Flow Reactor (PFR)	Space Time Option
CSTR	Space Time Option
CSTR	Dead Space Option
CSTR	Initialization from Stream
Data Recorder	

7.1.2 Saving a Simulation

HYSYS has three different save commands:

Command	Description
Save	Saves the case using the current file name and location.
Save As	With this saving option, you need to enter a name and location to which you want to save the file. The **Save Simulation Case As** view appears when you select the **Save As** command. You are able to choose the **File Path** and a **File Name** for the case. HYSYS automatically attaches the appropriate file extension, ***.hsc***.
Save All	Use this option to save all currently opened HYSYS cases. You are asked to select which cases should be saved. Select multiple cases with the **CTRL** key and the primary mouse button. Once the cases have been highlighted, select the ***Save*** button to save the case(s) as shown in the view, or the ***Save As*** button to save with a new name and/or location.

Figure 7.4

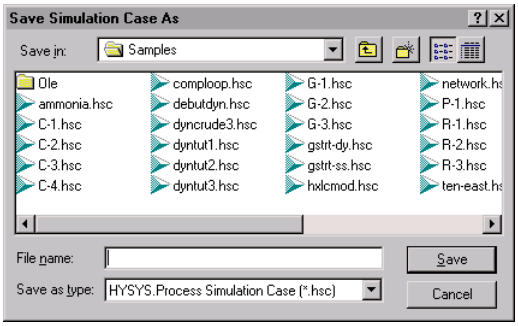
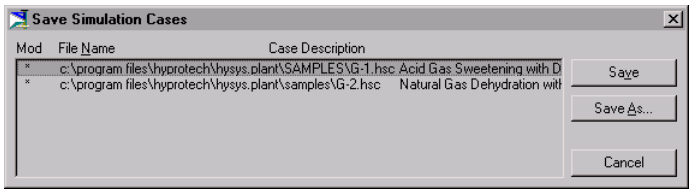


Figure 7.5

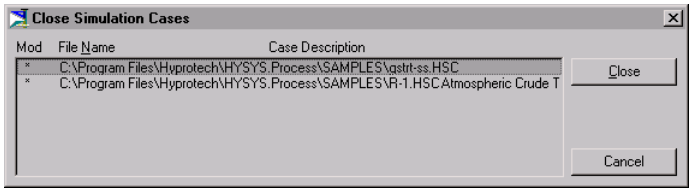


7.1.3 Closing a Simulation

Two of the options under **File** in the Menu Bar involve closing your simulation:

Command	Description
Close Case	Closes the active case. Before closing the case, you are asked if the case should be saved.
Close All	Allows you to close more than one case at a time. The name of each opened file is displayed. You select which case(s) you want to close.

Figure 7.6



7.1.4 Printing

The printing section contains the following options:

*See **Section 6.1 - Printing in HYSYS** for more information.*

Command	Description
Print	Allows you to print Specsheets for streams and operations.
Print Snapshot	Prints a bitmap snapshot of what currently appears in the active HYSYS view.
Printer Setup	Allows you to select the default printer, print orientation, paper size, etc. It is similar to the Printer Setup commands in other Windows applications.

7.2 Edit

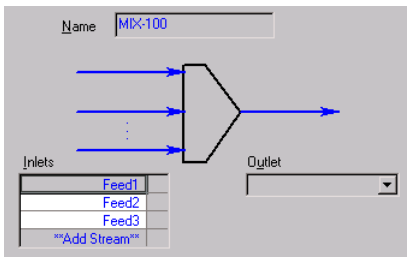
Figure 7.7

The functions listed under **Edit** in the Menu Bar are available in all Environments (i.e. Basis, Main, Sub-Flowsheet) and can be used both across the Environments and outside of HYSYS:

Command	Description
Cut	Removes the selected cell(s) from the current view. You can then use the Paste function to place the removed cell(s) in another location or in another application.
Copy	Copies the selected cell(s) to the Clipboard. You can then use the Paste function to place the copied cell(s) in another location or in another application.
Copy Special / with Labels	Copies the selected cell(s) and their corresponding labels to the Clipboard. You can then use the Paste function to place the copied cell(s) in another location or in another application.
Paste	Allows you to place copied or cut selections in the location of your choice.

Although the functions are available throughout HYSYS, not all areas within the Environments support them. Only matrix type areas can be manipulated. For instance, you can copy a group of cells from the Workbook or from a table and paste them into the Spreadsheet. In most areas, you will be able to manipulate a group of cells by clicking and dragging with the mouse. Whenever cells are grouped within a matrix (i.e. Inlet streams for a **MIXER**, shown in Figure 7.8), you can highlight more than a single cell.

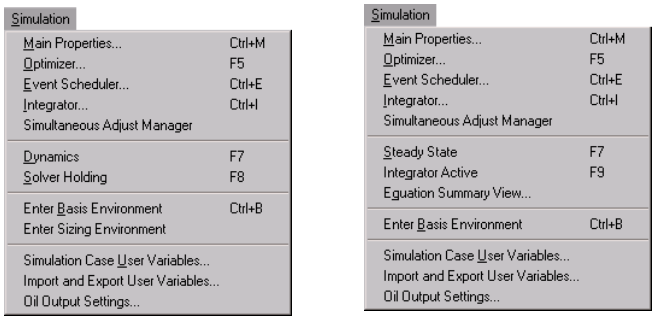
Figure 7.8



7.3 Simulation

The options under **Simulation** vary depending on whether you are in Steady State or Dynamic mode. Both steady state and dynamic views are shown below.

Figure 7.9



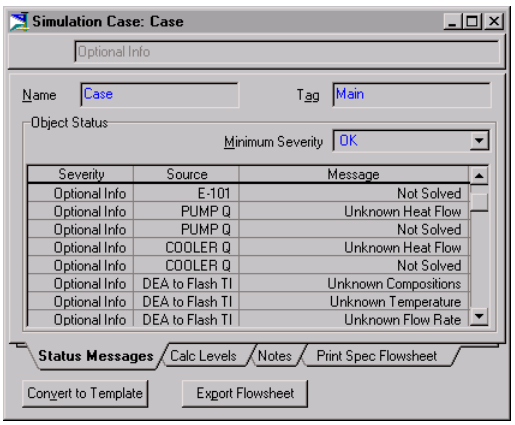
Option	Description
Main Properties	Displays simulation case properties, such as Calculation Levels, Case Description and current Object Status messages.
Optimizer	Opens the Optimizer, which performs steady state optimization. This option is available from the Main Flowsheet only.
Event Scheduler	Opens the Event Scheduler. Although it appears in the **Steady State** menu, it only functions when in Dynamic Mode.
Integrator	Opens the Dynamic Integrator. Although it appears in the Steady State menu, it only functions when in Dynamic Mode.
Simultaneous Adjust Manager	Allows you to monitor and modify all adjusts that are selected as simultaneous. Refer to **Section 10.1 - Adjust** for more information.
Dynamics/Steady State	Toggles between Dynamic and Steady State. The **Simulation** menu is redrawn depending on this choice.
Solver Holding/ Solver Active	Toggles between Hold mode and Calculate mode. This option only appears in Steady State. When HYSYS is calculating, the **Hold Calculations** option appears, and when in holding mode, the **Start Calculations** option appears.
Start Integrator/Stop Integrator	Toggles between Start Integrator and Stop Integrator. This option only appears in the **Simulation** menu when you are in Dynamic mode.
Equation View Summary	Opens the **Equation Summary** view.

Option	Description
Enter Basis Environment	Opens the Simulation Basis Manager.
Simulation Case User Variables	Opens the **User Variables** view for the current case.
Import and Export User Variables	Opens the **Import and Export User Variables** view allowing you to export user variables from the current case to an export file and import user variables from an export file to the current case.
Oil Output Settings	Opens the **Preferences** view to the **Oil Input** tab, where Assay preferences can be set for the case.

7.3.1 Main Properties

Selecting the **Main Properties** option via **Simulation** in the Menu Bar or the CTRL M hot key opens the **Simulation Case** view, as shown in Figure 7.10.

Figure 7.10



You can convert the case to a template by selecting the ***Convert to Template*** button at the bottom of the view. This converts the entire Flowsheet to a Template.

You can also export all the information on the **Print Spec Flowsheet** tab to a text file using the ***Export Flowsheet*** button. HYSYS will open a dialog box to save the information to a text file with an *.hsi extension.

Main Environment Button

If the Main Properties option is selected while in a Sub-Flowsheet environment, a ***Main Environment*** button is available to return to the Main environment.

Status Message Tab

You can use this tab is used to re-name the environment. Notice that the **Name** cell displays the name **Case**, which is the internal name applied to the simulation case. This name appears in the upper right hand corner of the HYSYS screen when in the Main Environment. It also appears as part of the name on the Workbook and on the PFD. The **Tag** cell displays the word **Main**, which also appears in the upper right hand corner of the HYSYS screen and as part of the Workbook and PFD names. You can change either of these descriptive words.

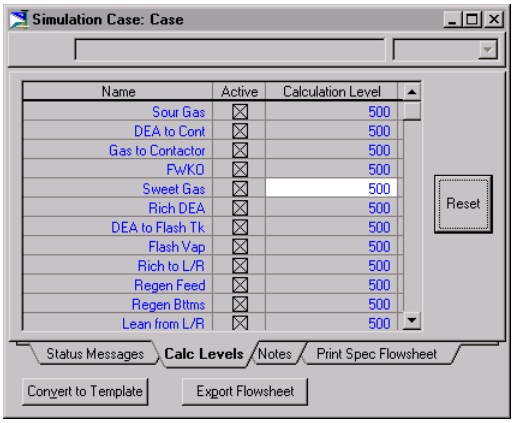
*See **Section 5.4 - Object Status Window/Trace Window** for more details on choosing a Minimum Severity.*

The **Object Status** group displays the current status messages for all objects in the Flowsheet according to the **Minimum Severity** that you choose.

Calculation Levels Tab

The second tab of the **Simulation Case** view is the **Calc Levels** tab. The Calculation Level sets control over the order in which streams, operations, and Flowsheets are calculated. For instance, you may want a certain operation (Adjust) to be performed before another (Recycle). To do this, you specify the Calculation Levels, such that the Adjust has a lower value than the Recycle. **HYSYS then forces the operation with the lower Calculation Level to calculate first**.

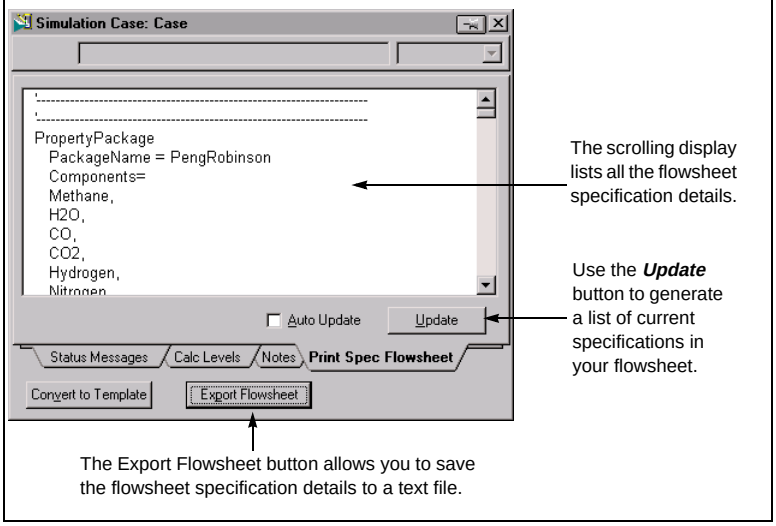
Figure 7.11



You can also ignore a certain stream or operation in a case by deactivating its **Active** check box. The ***Reset*** button will return all the **Calculation Levels** to their default values.

Print Spec Flowsheet Tab

Figure 7.12



Selecting the Auto Update checkbox, will automatically update the list whenever the simulation changes.

The last tab of the **Simulation Case** view is the **Print Spec Flowsheet** tab. Once updated, this view lists all the current specifications used to build your simulation case. The display begins with the Basis Environment information and then proceeds to the Simulation Environment, listing all the specifications for the unit operations attached to your case.

To print the information displayed on the **Print Spec Flowsheet** tab to a file, select the ***Export Flowsheet*** button at the bottom of the view. The **Save PrintSpecFlowsheet File** dialog box will appear which will allow you to save this information as a text file with the extension *.hsi.

7.3.2 Optimizer

The Optimizer can be invoked with the hot key ***F5****.*

The Optimizer performs steady-state optimization by finding values of process variables that minimize or maximize a user-defined objective function. The Optimizer possesses its own Spreadsheet, where the variables that define the objective function are attached, as well as the mathematical expression relating these variables. In addition, you may set upper and lower bounds and constraints that the process variables must also satisfy. The optimization must begin in a feasible region, i.e., all constraints must be satisfied at the initial starting conditions.

7.3.3 Event Scheduler

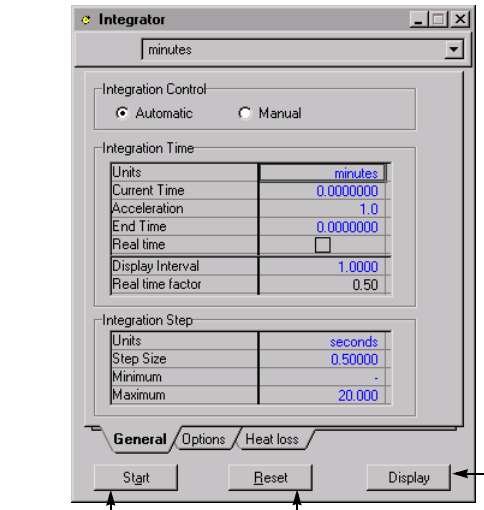
Using the Event Scheduler it is possible to have HYSYS perform tasks at given times during the dynamic simulation of a process. The time at which HYSYS starts a task may be predetermined (i.e. after 20 minutes of simulation time) or it may depend on the simulation (i.e. after a reboiler product stream temperature stabilizes).

Refer to Chapter 2 - Dynamic Tools of the **Dynamic Modeling** manual for more information regarding the Event Scheduler.

7.3.4 Integrator

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

Figure 7.13



Automatically forces the Flowsheet Variables to update regardless of what is set for the Display Interval.

Starts the Integrator. When the Integrator is running, this switches to a ***Stop*** button. The Start/ Stop functionality can also be accessed by using the Integrator buttons in the Button Bar.

Resets all the values to the default values.

On the **Integrator** property view, you can specify various integration parameters:

Parameter	Description
Units	Select the time units for the Current Time, End Time and Display Interval.
Current Time	Displays the time that the Integrator has been running. When the Integrator is Reset, this value returns to zero.
Acceleration	Allows you to see the Real Time factor to slow a simulation case down to run at real-time.
End Time	Allows you to specify the time at which the integrator will stop.
Display Interval	Time interval at which HYSYS updates the variables in the Flowsheet. Keep in mind that the frequency of update has a significant impact on the speed at which your simulation runs. The Display Interval has no affect on the calculation frequency.
Units	Select the time units for the Step Size, Minimum and Maximum cells.
Step Size	When the Integrator starts, the Step Size varies according to the integration errors and the range set by the Minimum and Maximum. After the ***Reset*** button is pressed, HYSYS defaults this value to the Minimum specified step size.
Minimum	Allows you to specify the minimum step size.
Maximum	Allows you to specify the maximum step size.

7.3.5 View Equations

The View Equations command is available only when you are in Dynamics mode.

The Equation Summary View is used when running a case in Dynamic mode. The Equation Summary View is opened automatically by HYSYS when there are dynamic specification errors in your case, or manually by selecting **View Equations** from the **Simulation** Menu Bar. The Equation Summary provides a list of the equations and pressure flow specifications that are currently used in the dynamic simulation. In addition it allows you to perform an analysis of the simulation to determine whether any equations and specifications are required or redundant.

7.3.6 Enter Basis Environment

You can use the hot key CTRL B to re-enter the Basis Manager from any Environment.

When in the Basis Environment, you can access the Basis Manager via the **Basis** command in the Menu Bar. Whenever you open a New Case, HYSYS places you directly into the Basis Manager. You can re-enter the Basis Manager from any Flowsheet by choosing **Simulation** from the

Menu Bar and then the **Enter Basis Environment** option. You can also use the ***Basis*** button to re-enter the Basis Manager from either the Main or Column environment.

The **Simulation Basis Manager** is a Property View that allows you to create and manipulate every fluid package in the simulation. Each Flowsheet in HYSYS can possess its own fluid package. Because the Template and Column Sub-Flowsheets reside inside the Main Simulation, these Sub-Flowsheets can either *inherit* the fluid package of the Main Flowsheet, or you can create an entirely new fluid package for each Sub-Flowsheet. For each Fluid Package you can supply a property package, components, reactions, and user properties.

*Refer to **Section 1.3 - Starting a Simulation** for more information.*

7.3.7 Oil Output Settings

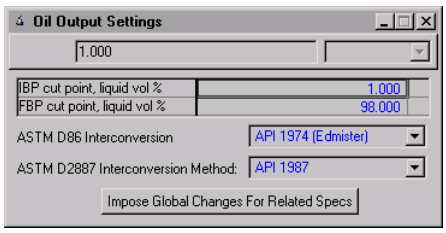
During the characterization of an oil or the calculation of boiling ranges for a fluid in the BP Curves utility, the initial (IBP) and final boiling point (FBP) cut point values can have a significant effect on the outer limits of the boiling range. During the laboratory analysis of a petroleum fluid boiling point regions, the most difficult to measure are the end regions.

*Refer to **Chapter 3 - HYSYS Oil Manager** in the **Simulation Basis** manual for information on oil characterization.*

*More detail on the **BP Curves** utility can be found in **Chapter 8 - Utilities**.*

HYSYS uses the defaults of 1% and 98% for the IBP and FBP, respectively. With a 1% IBP value, HYSYS utilizes the boiling points of all components in the first volume percent of the given fluid and calculates a weighted average boiling point, which is used as the IBP for any further analysis. The final boiling point is determined in much the same way, using the weighted average of the boiling points for the components found in the final two liquid volume percent of the fluid.

Figure 7.14



On the **Boiling Ranges** view, you can specify the following, in units of liquid volume percent:

Unit	Description
IBP cut point	Values from 0% to 5% are allowed.
FBP cut point	Values from 90% to 100% are allowed.

Also, you can choose the **ASTM D86 Conversion Method**. The options include:

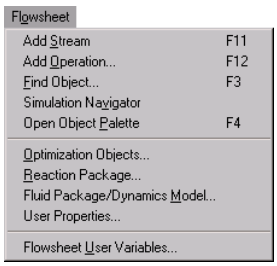
- API 1974
- API 1987
- API 1994
- Edmister-Okamoto 1959

For the **ASTN D2887 Conversion Method**, the options include:

- API 1987
- API 1994 Indirect
- API 1994 Direct

7.4 Flowsheet

Figure 7.15



The options under **Flowsheet** are:

Option	Description
Add Stream	Adds a Stream to the Flowsheet. For more information, refer to **Section 1.3.4 - Installing Streams**
Add Operation	Adds a Unit Operation to the Flowsheet. For more information, refer to **Section 1.3.5 - Installing Operations.**

Option	Description
Find Object	Opens the Object Navigator and allows you to access any stream, unit operation or Flowsheet present in the case. Refer to **Section 5.2.1 - Object Navigator** for more information.
Simulation Navigator	Opens the Simulation Navigator that allows you to view the Property View of any stream or unit operation present in the case. Refer to **Section 5.1.3 - Simulation Navigator** for more information.
Open/Close Object Palette	Toggles between opening and closing the Object Palette. Refer to **Section 1.3.3 - Object Palette** for more information.
Optimization Objects	Allows you to select a generic set of objects which are used to identify the underlying flowsheet variable, and provide the necessary configuration information for use by Optim or Estim. Refer to **Section 3.2 - Optimization Objects** in the RTO manual.
Reaction Package	Accesses the Reaction Package view.
Fluid Package/ Dynamic Model	Accesses both the **Fluid Packages** assignment view, and the Dynamic Property Model.
User Properties	Accesses a view from which you can install **User Properties**.
Flowsheet User Variables	Accesses the User Variables view. For more information, see **User Variables** chapter of **Customization Guide**.

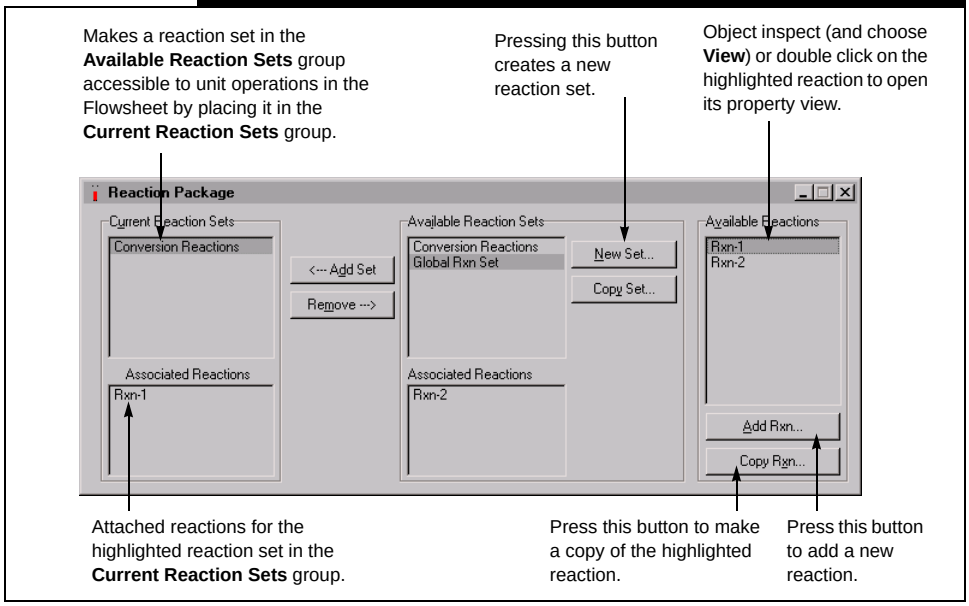
7.4.1 Reaction Package

*Components for the reactions are installed in the Basis Environment. Refer to **Chapter 4 - Reactions** in the **Simulation Basis** manual for details on component selection.*

Selecting this option opens the **Reaction Package** view, from which you can do the following:

- Create, Copy or Edit a Reaction.
- Create, Copy or Edit a Reaction Set.
- Attach Reactions to a Reaction Set.
- Make a Reaction Set available to unit operations within the current case.

Figure 7.16



The **Reaction Package** view eliminates the need to return to the Basis Environment to define Reactions and Reaction Sets. The only aspect of defining reactions that must be done in the Basis Environment is the selection of components.

Generalized Procedure

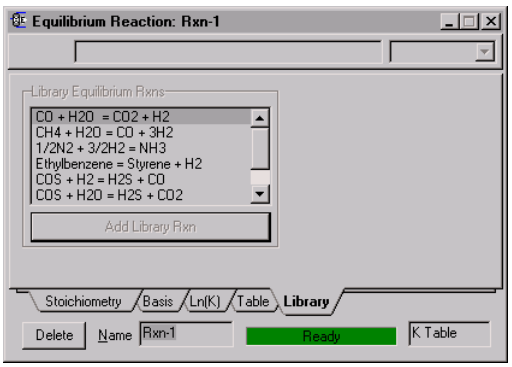
*Refer to Chapter 4 - Reactions in the **Simulation Basis** manual for details on Reactions and Reaction Sets.*

The following procedure outlines the basic steps for creating a reaction, creating a reaction set, adding the reaction to the reaction set and then making the set available to the Flowsheet. Refer to the **Reaction Package** view, shown in Figure 7.16, as you follow the procedure:

1. Select **Reaction Package** under **Flowsheet** in the Menu Bar.
2. On the **Reaction Package** view, press the ***Add Rxn*** button to create a new Reaction.
3. A **Reactions** view appears, from which you must select the type of reaction to create. Highlight a reaction type and press the ***Add Reaction*** button.

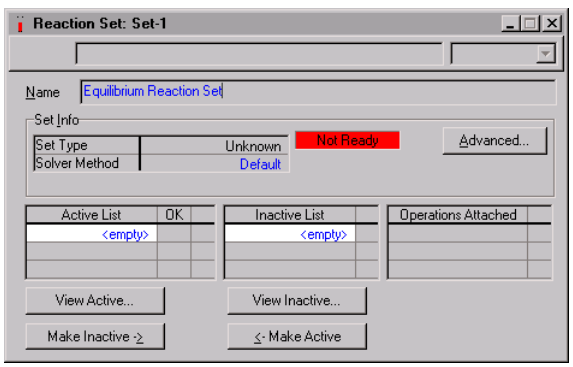
4. The property view for the reaction type you have selected will appear. Complete the input for the reaction until **Ready** appears as its status message. You can close the Reaction property view, if desired, by pressing the ***Close*** button.

Figure 7.17



5. On the **Reaction Package** view, press the ***New Set*** button to create a Reaction Set. The **Reaction Set** view appears.

Figure 7.18



6. If desired, change the **Name** of the Reaction Set to better identify it.
7. To attach the newly created reaction to the Reaction Set, place the cursor in the **<empty>** cell of the **Active List** group. Open the drop down list in the Edit Bar and select the reaction name. The reaction becomes attached to the Reaction Set, as indicated by the activated check box in the **OK** column.
8. Press the ***Close*** button on the **Reaction Set** view.

9. In the **Available Reaction Sets** group of the **Reaction Package** view, highlight the name of the newly created Reaction Set. Notice that the attached reaction is listed in the **Associated Reactions** group.
10. Press the ***Add Set*** button to make the Reaction Set, and thus the Reaction, available to unit operations in the Flowsheet. The new Reaction Set is displayed in the **Current Reaction Sets** group.

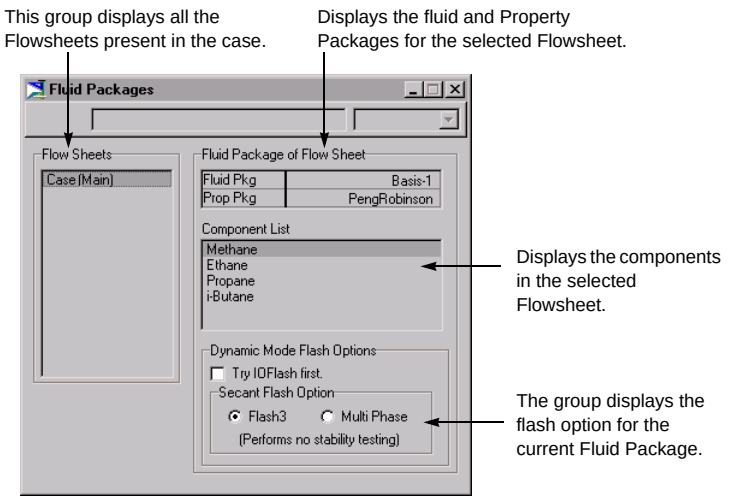
Add Set Button

7.4.2 Fluid Packages

If Flowsheets use the same fluid package, they must also use the same Flash Options.

This option quickly provides you with a summary of any Fluid Packages in the simulation. You can view the Property Package and components contained in each Flowsheet. Since each Flowsheet can have a different fluid and property package, each can also have a different Flash Option.

Figure 7.19



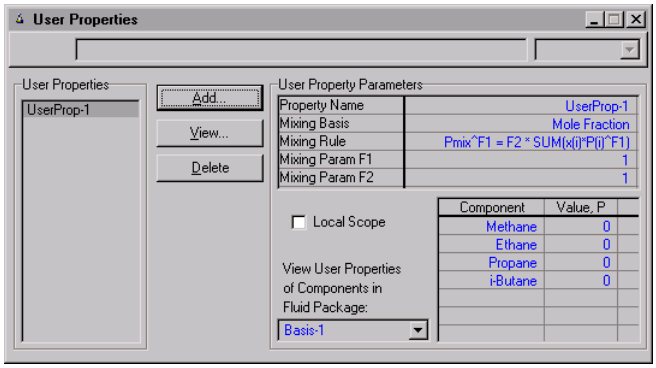
*For more information regarding the Dynamic Flash options refer to **Section 1.4.5 - Stability Test Tab** of the Simulation Basis Guide.*

7.4.3 User Properties

*Refer to **Chapter 5 - Component Maps** in the **Simulation Basis** manual for more information.*

The view accessed by selecting **User Properties** under **Flowsheet** in the Menu Bar allows you to manipulate user properties for your simulation case without returning to the Basis Environment. From this view, you can create, edit or delete user properties. Your changes are, however, transferred to the User Property view in the Basis Environment.

Figure 7.20



The Available User Properties are listed in the **User Properties** group. You can manipulate the User Properties by using the buttons on the view:

Button	Description
Add	Creates a new User Property.
Delete	Erases the currently highlighted User Property. HYSYS will not prompt for confirmation when deleting a User Property, so be careful when you are using this button.
View	Allows you to edit the currently highlighted User Property.

The **User Property Parameters** group lists the parameters for the highlighted user property.

You can make changes to the parameters directly on this view:

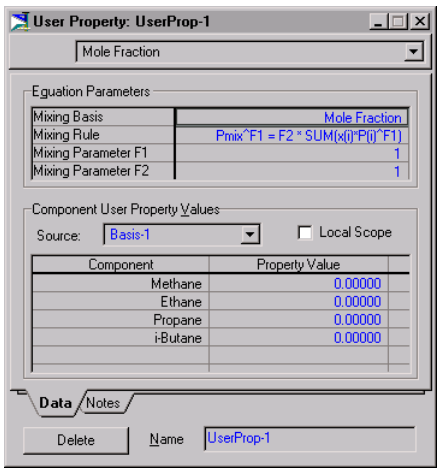
Parameter	Description
User Property Name	Use the HYSYS default, or supply the name.
Mixing Basis	Select fraction or flow on a molar, mass or liquid volume basis.
Mixing Rule	Choose one of the mixing rule equations: • $(Pmix)^{F1} = F2\sum_{i=1}^{N}(x(i)P(i)^{F1})$ • $\ln(Pmix)^{F1} = F2\sum_{i=1}^{N}(x(i)\ln P(i)^{F1})$ • $Index = \sum_{i=1}^{N} x(i)(F1P(i) + 10^{F2P(i)})$ *where:* P_{mix} - *total user property value* *P(i)* - *property value for component i* *x(i)* - *component fraction* *Index* - *total user property value for the index equation*
Mixing Parameter F1, F2	You can change either parameter from its default value of 1.
Component Property Values	Supply the component user property values.

From the **View User Properties of Components in Fluid Package** drop down list, select a Fluid Package as the source for the user property list of components.

Adding a User Property

1. Press the ***Add*** button.
2. The **User Property** view will appear as shown in **Figure 7.21**.

Figure 7.21

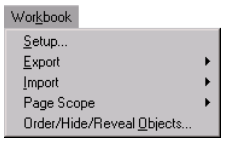


3. Provide a descriptive **Name** for the user property.
4. In the **Equation Parameters** group, select a **Mixing Basis** (Fraction or Flow): Mole, Mass or Liquid Volume.
5. Choose a **Mixing Rule**.
6. You can modify the two **Mixing Parameters (F1** and **F2)** to more accurately reflect your property formula.
7. Select a **Source** fluid package to obtain the list of user property components.
8. Input the property values for each component.

7.5 Workbook

Refer to Chapter 4 - Workbook for more information.

Figure 7.22



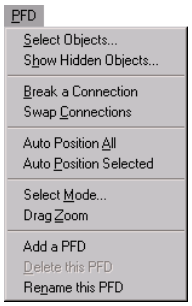
This menu item only appears when the Workbook has focus. The options under **Workbook** are:

Command	Description
Setup	Produces a dialog box from which you can manipulate Workbook tabs.
Export	Provides access to a submenu that contains the **Workbook** and **Pages** options. Allows you to save a Workbook or Page setup. A name, file path and description for the Workbook can be provided.
Import	Provides access to a submenu that contains the **Workbook** and **Pages** options. Replaces the current Workbook or Page setup with a saved setup.
Page Scope	Provides access to a submenu that contains the **Show/Hide Sub-Flowsheet Objects** option. This toggle function will either show or hide Sub-Flowsheet objects on the active Workbook tab.
Order/Hide/Reveal Objects	Produces a dialog box that you can choose to sort the Workbook objects either alphabetically or manually, hide Workbook objects and reveal Workbook objects.

7.6 PFD

Refer to Chapter 3 - PFD for more information.

Figure 7.23

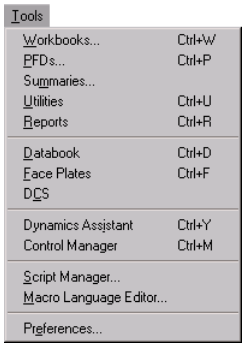


This menu item only appears when the PFD has focus. The options under **PFD** are:

Command	Description
Select Objects	Allows you to select multiple operations and streams on the PFD. The **Select Objects** dialog contains a Filter, allowing you to narrow your object choice.
Show Hidden Objects	Any hidden objects on the PFD may be viewed with this option. When you select this, HYSYS displays the **Show Hidden Objects** dialog.
Swap Connections	Allows you to select two streams which are attached to the same operation and exchange their nozzle connections.
Break a Connection	Lets you break the connection between a stream and an operation without deleting either.
Auto Position All	Repositions all objects on the PFD to the best possible location as selected by HYSYS.
Auto Position Selected	Auto positions only selected objects.
Select Mode	Selects the operating mode for the PFD. This is either Move, Size, or Attach.
Drag Zoom	Use the primary mouse button to drag a box around a region. HYSYS redraws the PFD showing only that region.
Add a PFD	Adds a new page to the PFD Notebook. The option to clone an existing PFD is presented.
Delete this PFD	Deletes the active PFD without a notice to confirm the action.
Rename this PFD	Allows you to change the name of the PFD, which appears on the page tab.

7.7 Tools

Figure 7.24



The options under **Tools** are:

Command	Description
Workbooks	Opens the Workbook for any Flowsheet. Refer to **Chapter 4 - Workbook**, for more information.
PFDs	Opens the PFD for any Flowsheet. Refer to **Chapter 3 - PFD,** for more information.
Summaries	Opens the Summary View for the current Flowsheet. Refer to **Chapter 1 - Interface,** for more information.
Utilities	Opens the **Available Utilities** view. Refer to **Chapter 8 - Utilities,** for more information.
Reports	Opens the Report Manager. Refer to **Section 6.2.1 - Creating a Report,** for more information.
Databook	Opens the Databook, from which you can install Process Data Tables, Strip Charts, Data Recorder Scenarios and Case Studies.
Face Plates	Provides access to controller Face Plates.
DCS	Accesses the **HYSYS DCS Interface**.
Dynamics Assistant	Opens the **Dynamics Assistant** to help convert steady state cases to dynamics.
Control Manager	The Control Manager lists all PID and MP controllers attached to your case. For more information refer to the **Dynamics Modeling** manual.
Script Manager	Provides access to the Script Manager.
Add-Ins Manager	Allows you to add Active-X controls to your HYSYS GUI.
Macro Language Editor	Accesses the HYSYS Macro Language Editor view.
Preferences	Accesses the User Session Preferences.

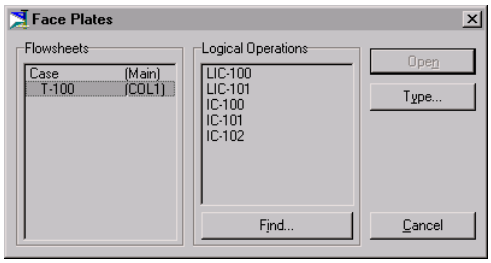
*The **User Session Preferences** view can be accessed only through the Tools menu.*

7.7.1 Face Plates

Refer to Section 10.7 - Controller Face Plate of the Dynamic Modelling Guide for more information.

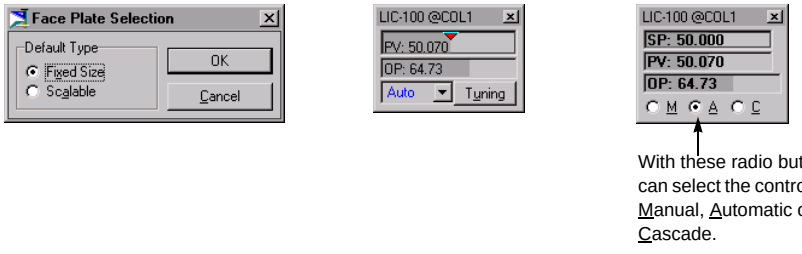
The Face Plate provides all pertinent information about a controller. You can access the **Face Plates** dialog under **Tools** in the Menu Bar or with the hot key CTRL F. From the **Flowsheets** group, select the Flowsheet containing the Controller whose Face Plate you want to open. The **Logical Operations** group displays all the Controllers in the highlighted Flowsheet. Select the Controller Face Plate(s) you want to open and press the **Open** button. You can select multiple Face Plates by holding down the CTRL key and clicking on the controller names.

Figure 7.25



You can change the appearance of all Face Plates in the Flowsheet by pressing the **Type** button. There are two types of Face Plates available in HYSYS, **Fixed Size** and **Scalable**. Fixed Size Face Plates are the default. Both types are shown below.

Figure 7.26



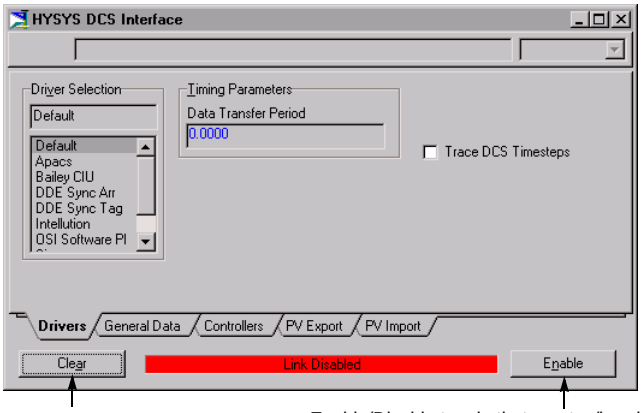
7.7.2 DCS

Selecting **DCS** under **Tools** in the Menu Bar opens the **HYSYS DCS Interface** view, from which you can link HYSYS to your Distributed Control System (DCS). Linking HYSYS to your DCS provides the following potential application opportunities:

- Conceptual Design
- Process Design
- Control Strategy Development
- Fault Tree and Safety Analysis and Evaluation
- Plant Control System Check Out
- Process Improvement
- Operator Training

*For further information on the **HYSYS DCS Interface**, contact your local AEA Technology Engineering Software representative.*

Figure 7.27



Deletes all DCS information from the current case. HYSYS asks for confirmation before the deletion.

Enable/Disable toggle that creates/breaks the communication link with the DCS.

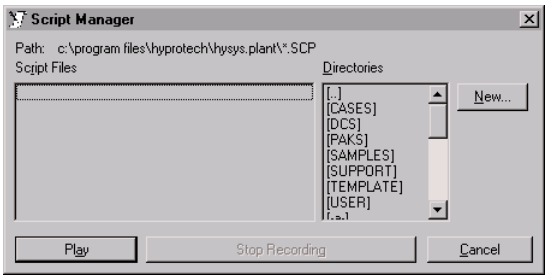
All information concerning the **HYSYS DCS Interface** is distributed over the four tabs on the view:

- Drivers
- General Data
- Controllers
- PV Export
- PV Import

7.7.3 Script Manager

The Script Manager is a HYSYS tool that records all your case interaction, with respect to installing streams or operations, making connections, and supplying specifications. The recorded script can be played back at a later time. When in the Basis, Main, or Column Environment, you can access the **Script Manager** under **Tools** in the Menu Bar.

Figure 7.28



When using the Script feature, it is important to note a few points.

- Session Preferences are not saved in the Script.
- Scripting is always done in HYSYS internal units.
- Scripting is Name specific, so stream and operation names in a script must be identical to those in the case in which you are running the script.
- For the playback of a script, the simulation case **MUST BE EXACTLY** as it was when the script was recorded, so that HYSYS can perform all the steps in the script.

Recording

The procedure for recording a New Script is as follows:

1. Save your simulation case. Since the case must be in exactly this same condition for playback of the recorded script, this is generally a good idea.
2. Select **Script Manager** under **Tools** in the Menu Bar.
3. Choose a directory from the **Directories** group in which the Script file will be saved.

4. Press the ***New*** button. HYSYS closes **the Script Manager** view and displays the **New Script** view.

Figure 7.29

5. Enter a **Name** and **Description** (if desired) for the Script. If you do not add an extension to the script **Name**, HYSYS adds the **.scp** extension.

Record Icon (Red)

6. Press the ***Record*** button to start recording. HYSYS closes the **New Script** view. Notice the red **Record** icon in the lower right corner of the DeskTop.
7. Perform each task that you wish to record.
8. When you finish recording commands, return to the **Script Manager** view and press the ***Stop Recording*** button.
9. If you would like to save the case, **DO NOT** save it with the same name as in step #1, as this will prevent you from playing back the script.

Playback

In order to play a script, the simulation case must be in the same state as it was prior to the recording of the script.

> At any time during the playback, you can stop the script by accessing the **Script Manager** view and pressing the ***Stop Play*** button. This stops the script, but may not stop the HYSYS function that was occurring during playback, i.e. integration in Dynamic mode.

Playback Icon (Green)

Follow this procedure to play a script:

1. Open the case that is associated with the script.
2. Select **Script Manager** under **Tools** in the Menu Bar.
3. Highlight the script name in the **Script Files** group of the **Script Manager** view. If your script is not listed in the default directory, you can select a different path in the **Directories** group.

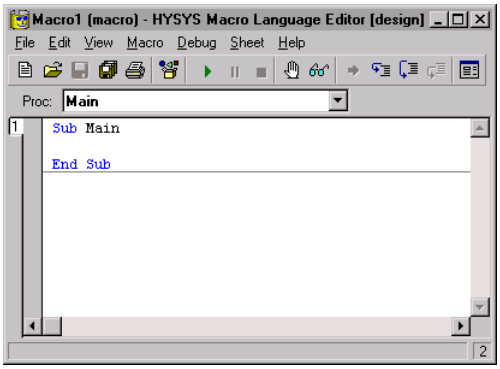
4. Press the ***Play*** button. HYSYS closes the **Script Manager** view and begins playing back the script. Notice the green **Playback** icon in the lower right corner of the DeskTop.
5. View the steps of the script playback in the Trace Window.

*Refer to **Section 5.5 - Object Status Window/Trace Window** for details on the Trace Window.*

7.7.4 Macro Language Editor

By selecting **Macro Language Editor** under **Tools** in the Menu Bar, you access the **HYSYS Macro Language Editor** view.

Figure 7.30



*For more information, consult the on-line help by selecting **Help** in the Menu Bar of the **HYSYS Macro Language Editor**.*

The HYSYS Macro Language Editor is an interactive design environment for developing, testing and executing WinWrap Basic scripts. The editor uses a syntax that is similar to Microsoft® Visual Basic®. See **Section 2.4 - Example 1 - The Macro Language Editor** in the **Customization Guide** for more information.

7.7.5 Preferences

The Preferences section in HYSYS is used to specify default information for the simulation case. This information includes Automatic Naming Formats, Units, Colours, Fonts, Icons, etc., for the simulation. Session Preferences can be saved for use in other simulations.

You can save multiple Session Preferences.

You can open the Session Preferences under **Tools** in the Menu Bar of the Basis, Main, and Column Environments. When opened, the Session Preferences Property View appears. Like all Property Views, there are a number of tabs associated with the view. Common to each tab are the two buttons along the bottom of the view:

Command	Description
Save Preference Set	Saves the Preferences to a file. You can provide a new file name and/or location to which the Preferences are saved.
Load Preference Set	This button allows you to Load Preferences saved from a previous HYSYS session.

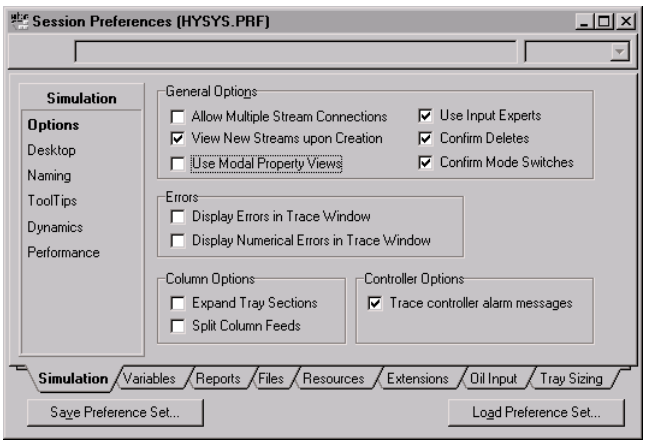
Simulation Tab

The Simulation tab consists of five pages: Options, Desktop, Naming, ToolTips and Assistant.

Options Page

The **Options** page shown in Figure 7.31 contains three groups: **General Options, Errors,** and **Column Options.**

Figure 7.31



The **General Options** group is a set of six check boxes:

Option	Description
Allow Multiple Stream Connections	This check box, which is not activated by default, controls whether lists of streams should be filtered to only those that are not currently connected. For example, suppose you use the drop down list of streams when selecting a feed to an operation. If this check box is not activated, only streams that are not already connected as a feed to an operation will be shown in the list. If the check box is activated, HYSYS displays all the streams in the Flowsheet, including the ones that you cannot connect as feed streams.
View New Streams Upon Creation	If activated, the property view for the stream will automatically appear when you add a new stream.
Use Input Experts	Column operations have an optional installation guide built in to assist you in the installation. When this check box is activated, you will be guided through the Column installation.
Confirm Delete	When this is activated, HYSYS prompts you for confirmation before deleting an object. If the check box is not activated, HYSYS will delete the object as soon as the instruction is given. It is advised to keep this option activated.
Use Modal Property Views	When this box is activated, all property views are displayed as Modal (with a Pin). If the check box is not activated, all property views are Non-Modal. When Views are Modal, you have the options of individually making each property view Non-Modal by selecting the Pin in the upper corner of the view.
Confirm Mode Switches	When activated, HYSYS prompts for confirmation when you change to or from Dynamic Mode.

*See **Section 1.1.8 - The Property View** for information on how to convert Modal views to Non-Modal.*

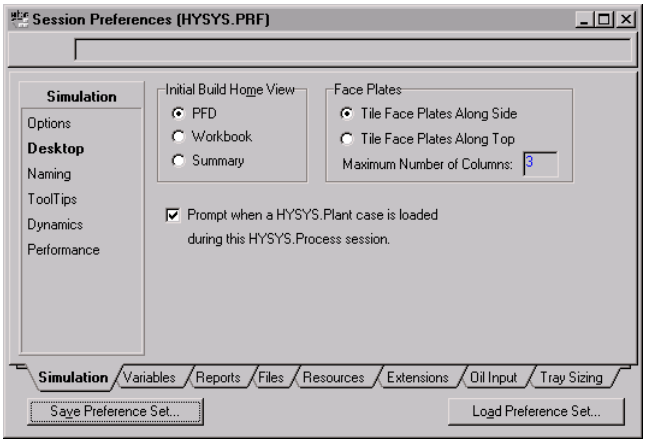
The **Errors** group contains two check boxes which when activated will send the specified errors to the **Trace Window**. When these check boxes are activated, you will not be prompted to acknowledge errors.

The **Column Options** group contains two check boxes. The **Expand Tray Sections** box will, when active, show a fully expanded column in the column environment PFD. When the check box is inactive, the column is displayed with the minimum required number of trays; those trays which have streams (inlet or outlet) attached to them.

Desktop Page

The **Desktop** page shown in Figure 7.32 contains two groups: **Initial Build Home View** and **Face Plates**.

Figure 7.32

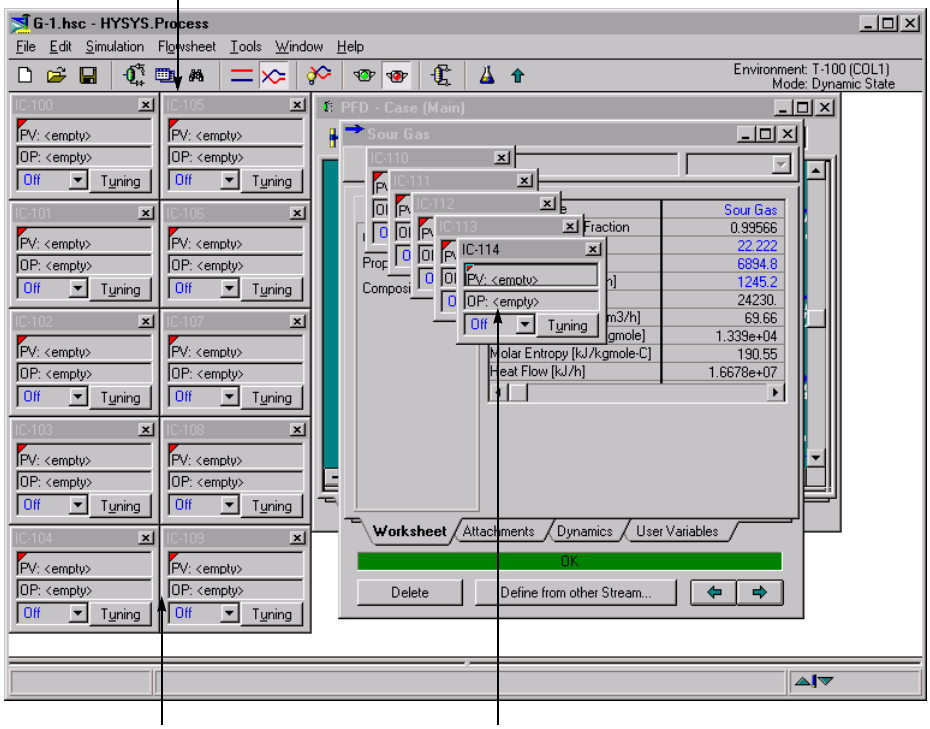


The **Initial Build Home View** group contains radio buttons which allow you to specify which of the three main interfaces, **Workbook**, **PFD**, or **Summary**, appears first when you enter a new Environment. This does not restrict you to what you can access within that environment, as the Workbook, PFD, and Summary Views can all be open at the same time. This setting only establishes which view should be opened by default.

The **Face Plates** group involves the placement of face plates on the Desktop. When you have a large number of face plates open in a case and you select **Arrange Desktop** under **Window** in the Menu Bar, the face plates will be organized according to your specifications in the **Face Plates** group. The face plates will either be placed along the left side of the Desktop in a column format or along the top in a row format. You can limit the number of columns or rows (depending on the choice of the radio button) in the **Maximum Number of Columns/Rows** input cell. Any excess face plates which cannot be placed in columns/rows due to the **Maximum Number of Columns/Rows** limit, will be cascaded with other open windows.

Figure 7.33

The **Tile Face Plates Along Side** radio button was selected on the **Simulation** page.



An input value of **2** was specified as the **Maximum Number of Columns**.

Extra windows, including face plates, are cascaded when **Arrange Desktop** is selected under **Window** in the Menu bar.

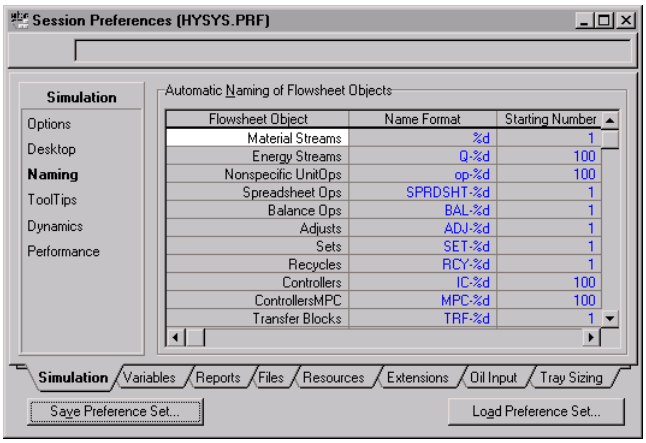
Naming Page

There are no restrictions in naming streams and operations. You can use more than one word (separated by spaces if desired) to provide the best possible description.

The **Automatic Naming of Flowsheet Objects** group dictates how HYSYS names streams and operations when they are installed. You can specify the naming convention for each type of operation. For each Flowsheet object, you can specify a naming convention and a starting number. For example, in the view shown, Energy Streams are indicated as **Q-%d**, with a Starting Number of **100**. The first energy stream installed using the **Add Energy Stream** command will be named **Q-100**, the second **Q-101**, and so on. The automatic naming function is

provided merely for convenience. You can change any default name at any time within the Flowsheet.

Figure 7.34

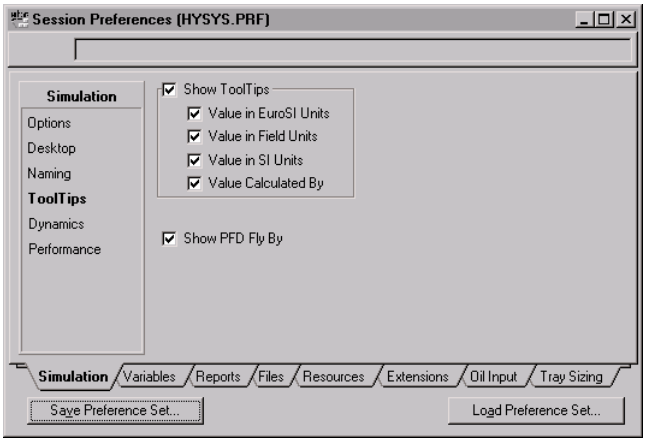


Tool Tips Page

There are two groups on the **Tools Tips** page, **Show Tooltips** and **Show PFD Flyby**. Tooltips show you information regarding objects such as streams. For example, make sure that all the check boxes in the **Show Tooltips** group are checked, and open the property view for a stream. Hover the mouse pointer over top of the temperature input cell. A tooltip appears, showing the units for the value and the calculation methods. If you only check the **Value in SI Units** check box in the **Show Tooltips** group, then when you hover the mouse pointer over top of the temperature input cell for a stream, the tooltip message would only show the temperature vaue in SI units.

A PFD flyby is a small message box that appears when you hover the mouse pointer over top of an icon on the PFD. If you uncheck the **Show PFD Flyby** box and then hover the mouse pointer over top of any icon on the PFD, the message box will not appear.

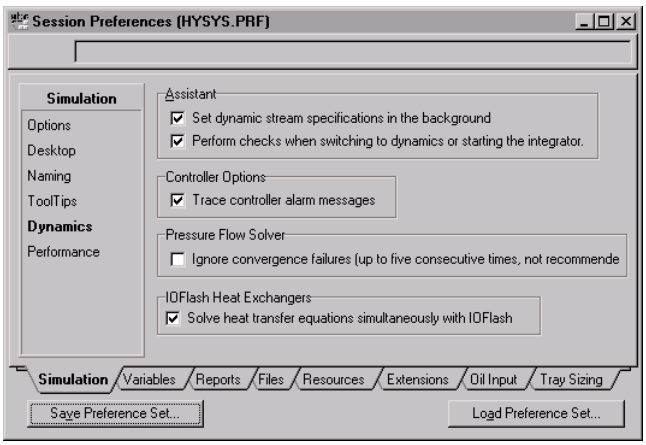
Figure 7.35



Dynamics Page

There are four groups on the **Dynamics** page that allow you to specify options used in Dynamic mode: Assistant, Controllers Options, Pressure Flow solver and IOFlash Heat Exchangers..

Figure 7.36



The Assistant group options deal with the Dynamic Assistant. The first check-box allows HYSYS to automatically set all of the dynamic stream

specifications. When the second check-box is active HYSYS will ask you if you would like to use the Dynamic Assistant when switching from Steady State to Dynamics or when starting up the Integrator.

The Controller Options group has one check-box that when active will display controller alarm messages in the trace window at the bottom of the screen.

The Pressure Flow Solver group allows you to ignore convergence failures for up to five pressure flow steps. It is not recommended to activate this option. However, it can be useful in extreme cases where you know that the case will converge but are having problems during one of the calculation steps.

The check-box available in the IOFlash Heat Exchanger group gives you the option of solving the heat transfer equations simultaneously with the IOFlash. This can be used to increases the calculation speed of your simulation. However, if you encounter any inconsistencies with your heat transfer equipment diable this option.

Performance Page

The **Performance Page** displays the **Macro Performance** slider. This allows you to balance the Macro speed with memory requirements. The faster your setting, the more memory your computer will require to run the macros.

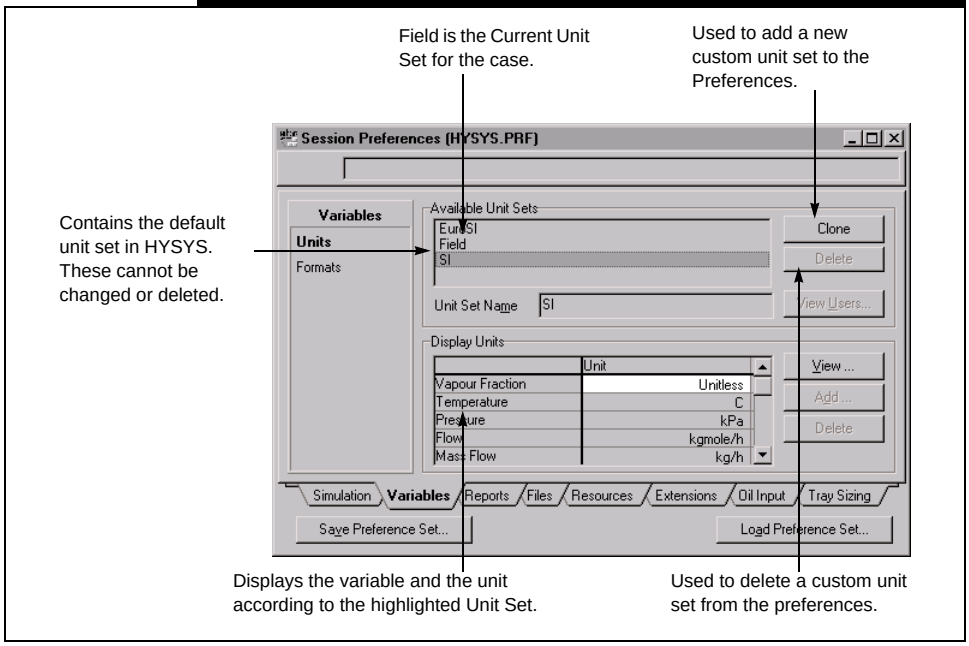
Figure 7.37

You also are given the option of updating the Object Status Services while solving.

Variables Tab

Units Page

Figure 7.38



Adding a New Unit Set

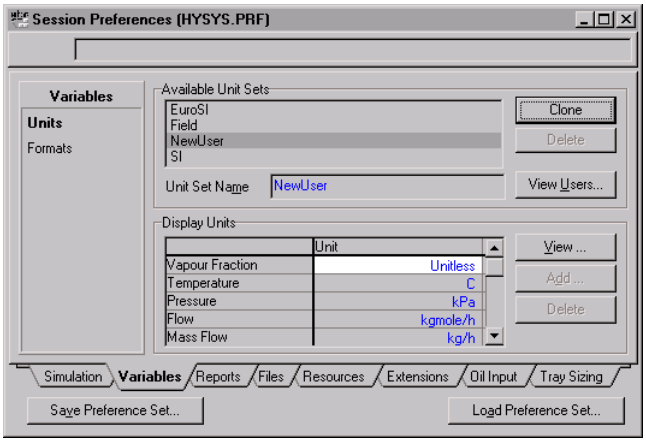
A custom unit set allows you to mix SI and Field units within the same unit set.

As previously mentioned, HYSYS has three default unit sets: **Field**, **SI**, and **EuroSI**. These three sets are fixed, in that none of the units can be changed. Since you may wish HYSYS to display information in units other than the default, you are able to create your own custom sets.

HYSYS allows you to create new custom unit sets by cloning an existing set and altering it. The procedure is as follows:

1. Select the unit set you want to modify from the **Current Unit Set** group. For example, choose **SI**.
2. Press the ***Clone*** button. The name HYSYS gives to this new unit set is **NewUser**. The name now appears in the **Current Unit Set** group and in the **Unit Set Name** cell. You can change the default name in the **Unit Set Name** cell. The units for the new set are the same as the SI units.

Figure 7.39



3. Move to the **Vapour Fraction** cell. Place the cursor on the unit cell which displays **Unitless**. If you click on the drop down list in the Edit Bar, it is empty because the chosen variable, Vapour Fraction, is unitless, and does not have any alternate units.
4. Move to the Temperature unit cell. It displays **C** (degrees Celsius), as its units. Open the drop down list in the Edit Bar. Now there are alternate units displayed: **C**, **K**, **F**, **R**. Choose **K** (Kelvin) as the base temperature unit. It appears in the Temperature Unit Cell.
5. Move to the Pressure Unit cell. It presently displays **kPa** as its units. Open the drop down list in the Edit Bar. There are several pressure units from which to choose. Use the scroll bar to select **psia**.
6. You can continue to scroll through the Variable/Unit list until you have finished customizing the Unit Set.
7. When you finish customizing the set, you can save it so that it will be available in future HYSYS cases. Press the ***Save Preference Set*** button. The Unit Set is stored as part of an overall Preferences (***.prf***) file.
8. Whenever you open a new case and wish to use your customized Unit Set, click on the ***Load Preference Set*** button, and then select the Preferences to load. Any customized Unit Set you saved within the Preferences appears as an **Available Unit Set**.

If you create a new set and then decide you do not want it, it can be deleted. Select the set you wish to delete from the **Current Unit Set** group and press the ***Delete*** button. The unit set is deleted from the current case.

Unit Conversion Buttons

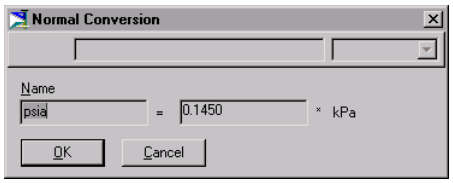
HYSYS performs its calculations in an internal unit set, and every other unit is converted from these default units. The three buttons on the right side of the Units page allow you to View, Add and Delete Unit Conversions.

Viewing a Unit Conversion

The ***View*** button lets you view the conversion factor HYSYS uses to convert from its internal unit (**SI**) to the unit chosen in your unit set. You can view the conversion for any available unit.

For instance, if you choose **Field** as the Unit Set, the pressure unit is **psia**. To see what conversion factor HYSYS uses to convert from psia into kPa (internal units), place the cursor on the pressure unit cell, and press the ***View*** button.

Figure 7.40



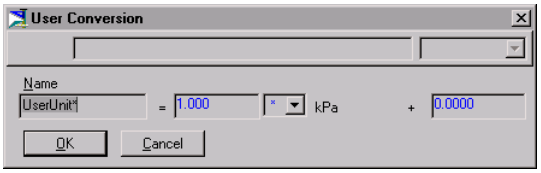
The **Name** cell displays the units, and the cell next to it shows the conversion factor (0.1450) between kPa and psia. This is a **View Only** display. You cannot change the contents of the **Name** cell or the value of the conversion for a default unit.

Adding a Unit Conversion

In some instances, you may require a unit that is unavailable in the HYSYS data base. In this case, HYSYS allows you to create your own unit and supply a conversion factor for it. You may only add a unit to a User defined Unit Set. The procedure for adding a Unit is as follows:

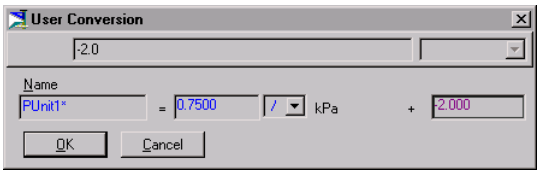
1. Select the customized **Unit Set** to which you want to add the unit.
2. Select the unit type (e.g., pressure).
3. Press the ***Add Unit Conv*** button. The **User Conversion** dialog appears.

Figure 7.41



4. By default, HYSYS names the new unit **UserUnit***. You can change this name. To avoid confusion, do not use a HYSYS default name. For this example, type **PUnit1** in the **Name** cell.
5. Enter the conversion factor between your unit and the HYSYS internal unit. Notice that the above view has two cells for entering values. The first is the multiply or divide cell, and the second is the add cell. The internal pressure unit in HYSYS is kPa, so the conversion will be from kPa. In the first cell, enter the value 0.75.
6. Next to the first cell is a cell containing an asterisk, indicating multiplication. The down arrow accesses a drop down list that also contains the division symbol (/). Select division as the operation.
7. The final cell is preceded by an addition symbol. The value in this cell can be added or subtracted. To subtract, place a negative sign in front of the number. For instance, use **-2.000** as the value for this cell.

Figure 7.42



8. Press the ***OK*** button. Notice the appearance of the name **PUnit1** in the Pressure unit cell.

The pressure conversion factor is now: **PUnit1 = 0.750/kPa - 2.000**.

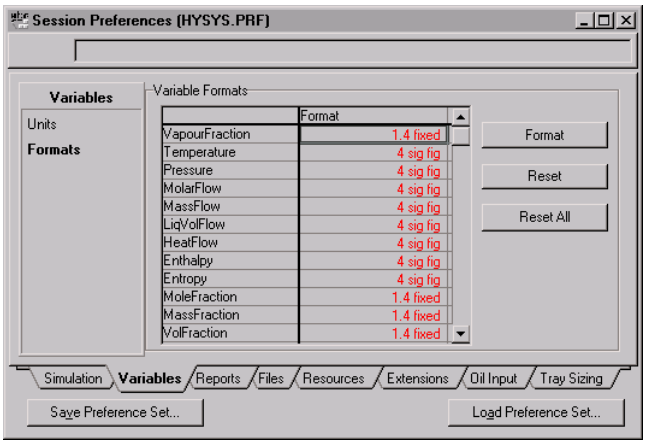
Deleting a Unit Conversion

You may only delete User defined Unit Conversions. Select the **Unit Set** which contains the unit to be deleted, then select the particular unit. For instance, in the customized unit set, **NewUser**, select **Pressure** as the unit to delete. Press the ***Delete Unit Conv*** button. The unit returns to the HYSYS default unit.

Formats Page

On this page you can specify how variables will be displayed. Double clicking on the format cell of a variable will bring up the **Real Format Editor** view.

Figure 7.43

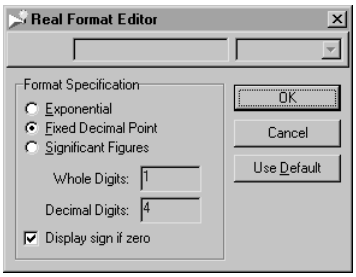


There are three buttons located on the **Formats** page:

Button	Description
Format	Brings up the Real Format Editor view for the active variable format cell.
Reset	Returns the selected variable to the default format.
Reset All	Returns all the variables to their default formats.

The **Real Format Editor** view is used to change variables format in HYSYS..

Figure 7.44



There are three radio buttons located in the **Format Specification** group box page:

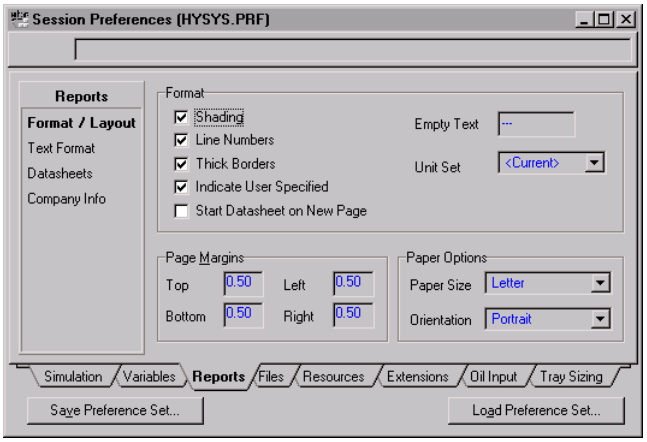
Radio Button	Description
Exponential	Displays the values in scientific notation. The number of significant digits appearing after the decimal point is set in the **Significant Figure** cell. As an example: • Entered or calculated value: 10000.5 • Significant Figure: 5 (includes the first whole digit) • Final display: 1.0001e+04
Fixed Decimal Point	Displays the values in decimal notation. The number of whole digits and significant digits appearing after the decimal point and are set in the **Whole Digits** and **Decimal Digits** cells. As an example: • Entered value: 100.5 • Whole digits: 3 • Decimal digits: 2 • Final display: 100.50 If the entered or calculated value exceed the specified whole digits, HYSYS will display the value as an Exponential, with the sum of the specified whole and decimal digits being the number of significant figures. Also, if the **Display sign if zero** check box is activated, HYSYS will display the sign of the number entered or calculated that has been rounded to zero.
Significant Figure	Displays the values in either decimal notation and scientific notation. The number of significant digits appearing after the decimal point is set in the **Significant Figure** cell. Example 1: • Entered or calculated value: 100.5 • Significant Figure: 5 • Final display: 100.50 Example 2: • Entered or calculated value: 10000.5 • Significant Figure: 5 • Final display: 1.0001e+04

Reports Tab

Format/Layout Page

This page provides you with options for formatting and specifying your reports.

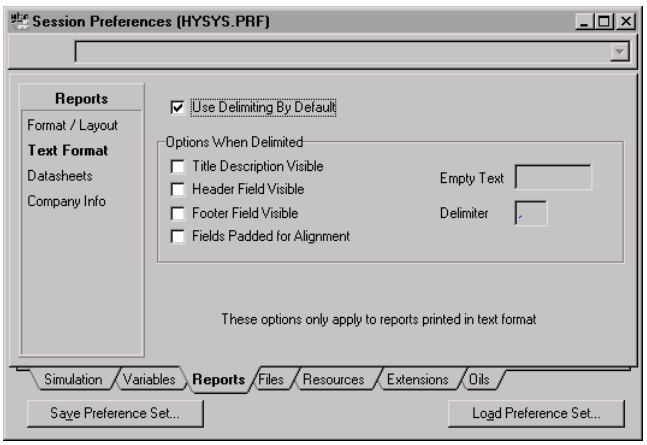
Figure 7.45



Text Format Page

For reports printed in text format, this page allows you to specify some text formatting options.

Figure 7.46

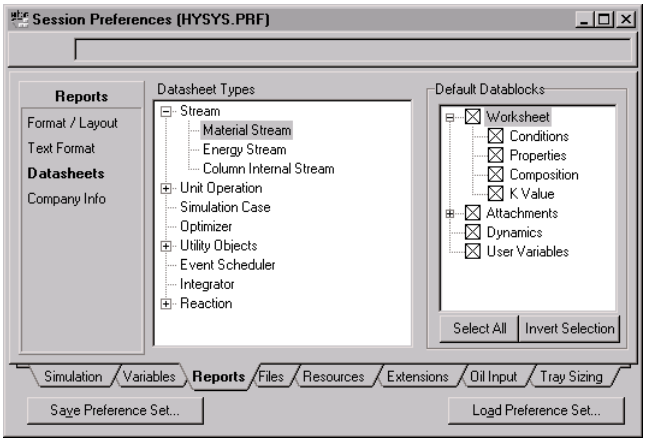


Datasheets Page

The **Datasheets** page allows you to select which datablocks are to be included for each stream, unit operation, utility and reaction report

printout. You first select the datasheet type in the **Datasheet Type** tree and then select the datablocks to be include or excluded in the **Default Datablocks** list box.

Figure 7.47

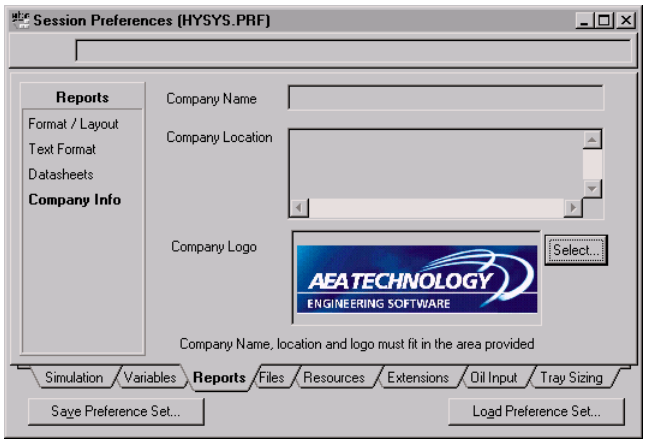


Company Info Page

On this page, you can provide some information about your company, such as the company name, location and logo.

Note that HYSYS will not automatically resize a bitmap file to fit the logo box on this page. The size of the sample logo shown in ***Figure 7.48*** *is* ***4.42cm*** *wide by* ***1.88cm*** *high. The maximum logo size that can be accommodated by the logo box is* ***6.55cm*** *wide by* ***2.38cm*** *high.*

Figure 7.48



Files Tab

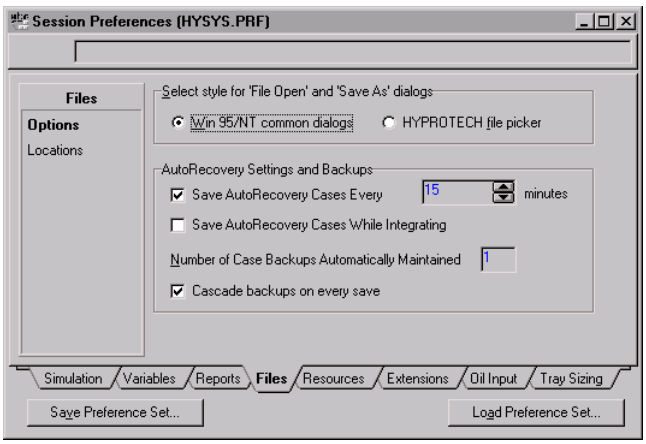
Options Page

The **Options** page lets you specify the view that will be used when opening or saving files in HYSYS. The default selection is the **Win 95/NT common dialogs** which will display the same open and save dialogs as other Windows applications. The **HYPROTECH file picker** selection displays the custom HYSYS open and save dialogs which show the build the case was saved in and the case description, if one was added to the case.

From the **Options** page there is the setting of where HYSYS saves AutoRecovery cases. Once this selection is activated, you can change the frequency of the saves from the default value of every 15 minutes. You can also set HYSYS to save AutoRecovery cases while the integrator is running.

There is also a box indicating the Number of Case Backups. HYSYS will maintain the specified number of backups of each simulation, using the extension *bk**. The newest backup will be *bk0*, the next newest *bk1*, etc.

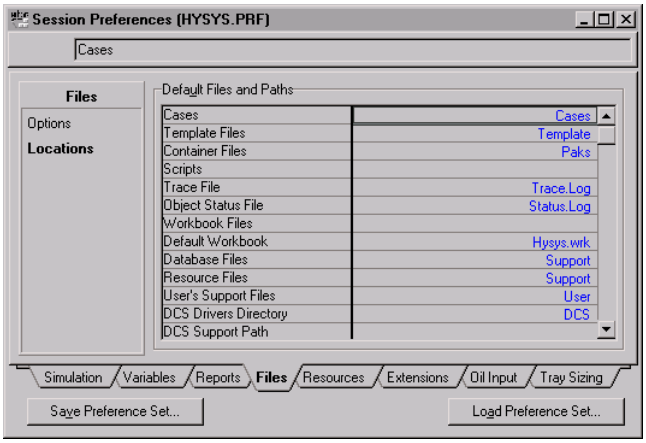
Figure 7.49



Locations Page

On the **Locations** page, you can select and specify the default paths for saving and reading case files.

Figure 7.50

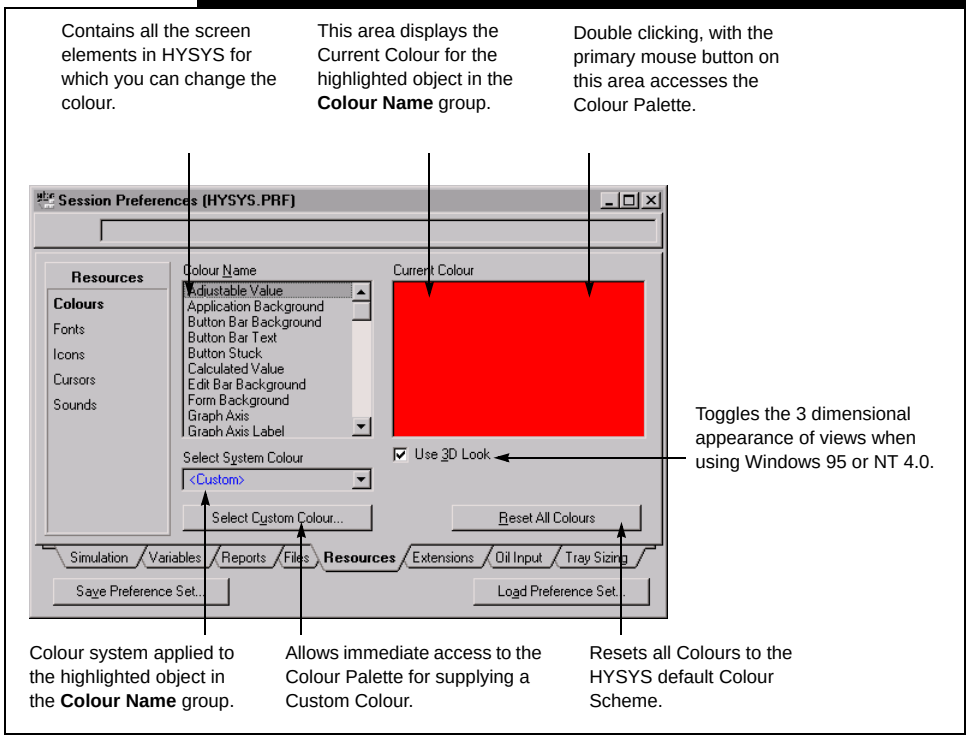


Resources Tab

Colours Page

All the functions and screen displays in HYSYS are set with a predefined colour scheme. However, you can make changes to this colour set and customize it to meet the specific needs of your simulation.

Figure 7.51



To change the colour of an element, you can use the following method:

1. Select the item from the **Colour Name** group. For instance, choose **Application Background**.
2. From the **Select System Colour** drop down list, select **<Custom>**.
3. Select the desired colour from the colour palette.
4. Press the ***OK*** button.
5. Notice that the Application Background has changed from the default colour to the new colour you selected. You can continue to change whatever item colours you wish.

*If you change any colour settings and then wish to return to the default HYSYS colours, press the **Reset All Colours** button.*

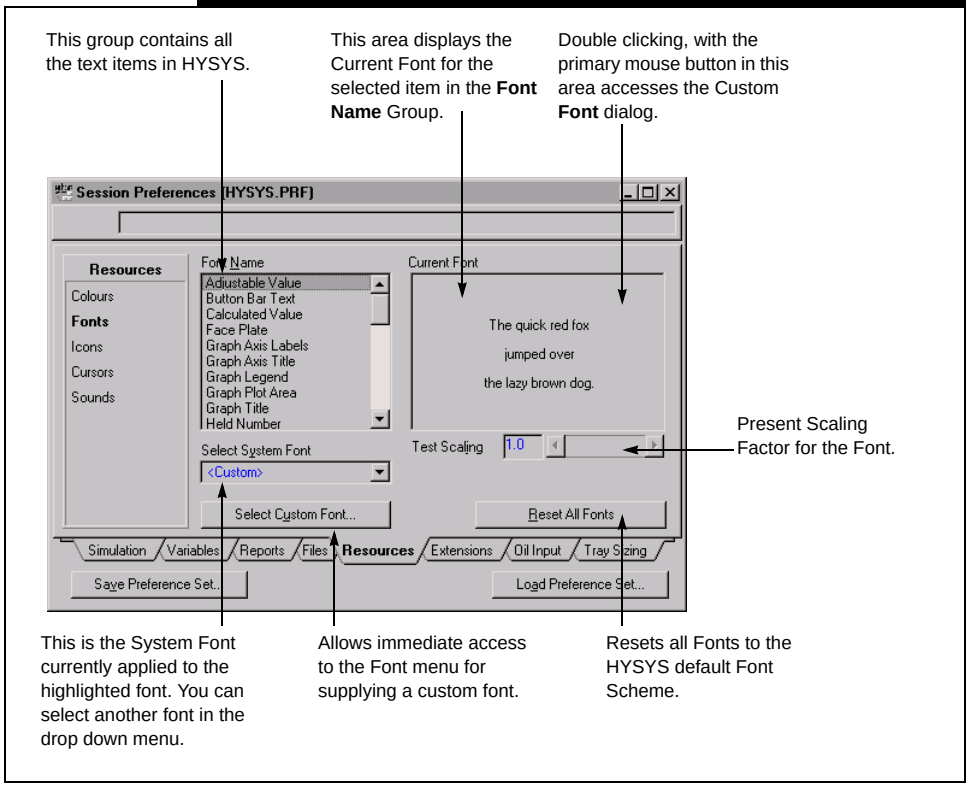
When you have finished customizing the item colours, you may want to save your preferences by pressing the ***Save Preference Set*** button.

Fonts Page

All the text appearing in HYSYS has a predefined font scheme. However, like the colour scheme, you are able to change the font scheme.

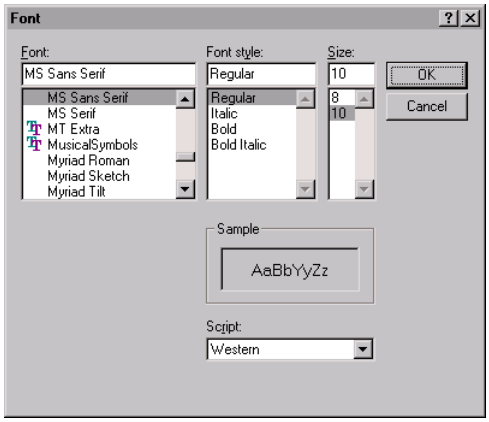
The structure of the **Fonts** page is much like the **Colours** page, as shown in the following view.

Figure 7.52



The ***Select Custom Font*** button opens the Custom **Font** dialog. From this view, you can see that the font used for a **Calculated Value** (the highlighted text item) is MS Sans Serif, Bold, 8 Point. You can change the font for this item by doing the following:

Figure 7.53



1. Select **MS Serif** from the **Font** group.
2. Select **Italic** from the **Font Style** group.
3. Select **10** from the **Size** group.
4. Press the ***OK*** button to return to the Preferences Font page. The new font appears in the **Current Font** group.

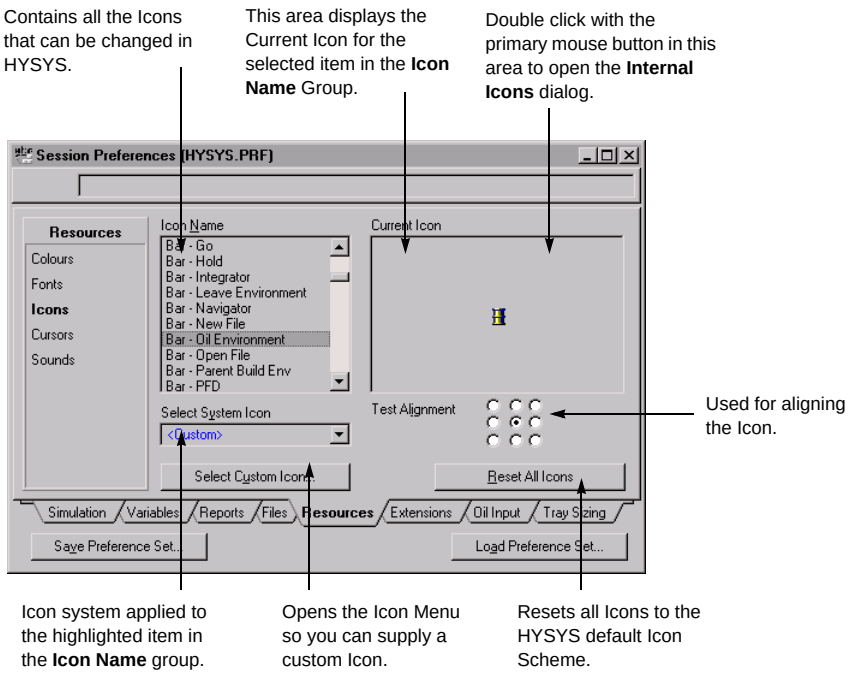
Notice the **Test Scaling** box below the **Current Font** box. This box is used for scaling the font. It presently displays 1.0. If you move the cursor to this box, you will be able to change this number in the Edit Bar. Increasing the number increases the size of the font, and decreasing the number decreases the font size.

When you have finished customizing the item fonts, you may want to save the scheme for future use by pressing the ***Save Preference Set*** button.

Icons Page

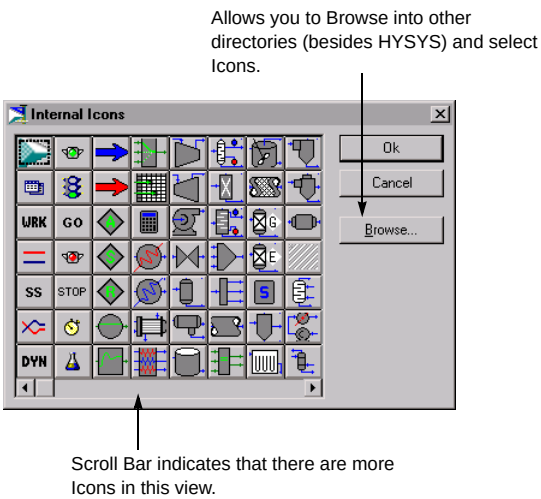
HYSYS has a default Icon scheme. Any Icons in HYSYS that you can change are displayed on this page. The structure of the **Icons** page is much the same as the **Colours** and **Fonts** pages.

Figure 7.54



In the above view, the highlight is on **Bar-Oil Environment**. The **Current Icon** box displays the current Icon used for this item. In the **Select System Icon** group, HYSYS displays **<Custom>**. This indicates that the Icon was chosen from the Custom Menu. If you choose the ***Select Custom Icon*** button, HYSYS displays the **Internal Icons** dialog. The **Internal Icons** dialog displays all the Icons available in HYSYS. You can choose any of these as the Icon for an item.

Figure 7.55



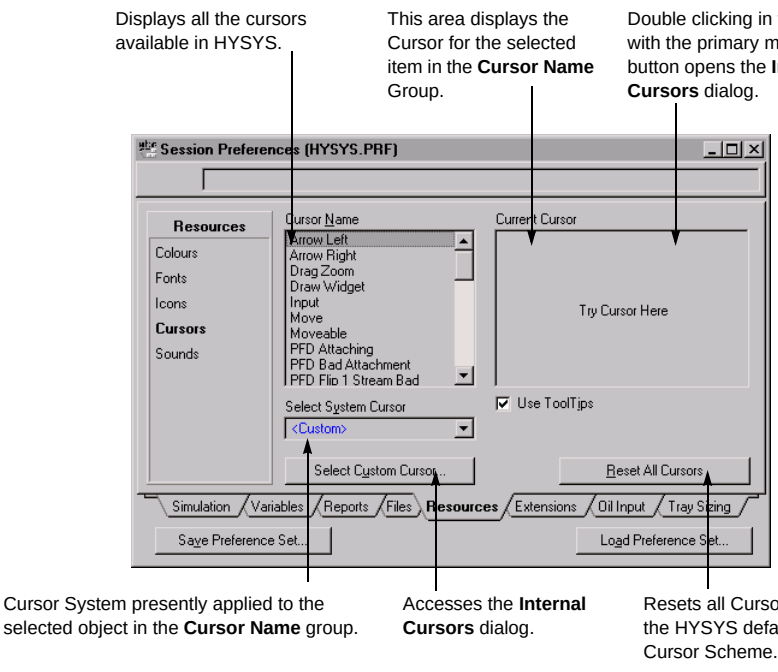
You can assign any Icon in this view to represent the highlighted item from the **Icon Name** group, in this case **Bar-Oil Environment**. If you choose to use the **Oil Drop** symbol as the Icon (column sixteen, row two), simply click on it and then click on the ***OK*** button. HYSYS returns to the **Icons** page view. The **GO** Icon will replace the **Exit** Icon in the Button Bar at the top of the screen.

When you have finished customizing the item Icons, you may want to save the scheme for future use by pressing the ***Save Preference Set*** button.

Cursors Page

As with the Colours, Fonts, and Icons pages, HYSYS allows you to customize the Cursors. Cursor type is often specific to certain functions. Any cursor types that you can change are displayed on the **Cursors** page.

Figure 7.56



If you scroll through the **Select System Cursor** drop down list, you can see the various cursor types available in HYSYS. At the very top of this list is the **<Custom>** type. If you click on this type, the **Internal Cursors** dialog appears. All Cursor types available in HYSYS are displayed. You can also access the **Internal Cursors** dialog by clicking on the ***Select Custom Cursor*** button.

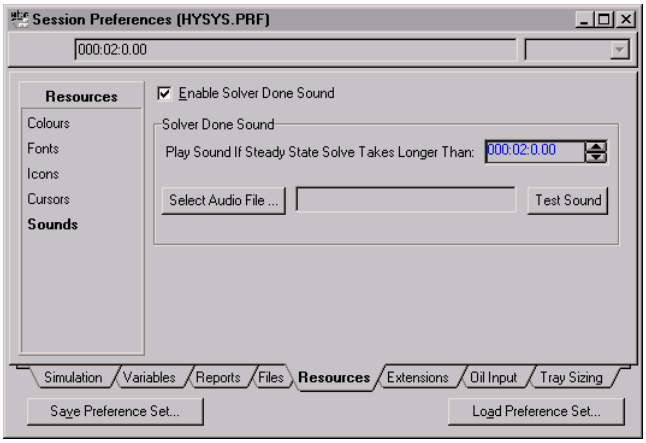
By selecting a Cursor type from the **Internal Cursors** dialog and pressing the ***OK*** button, you will change the cursor of the highlighted item in the **Cursor Name** group on the **Cursors** page. There is no view section on the **Internal Cursors** dialog, so you cannot view the cursor you are selecting until you return to the **Cursors** page.

When you have finished customizing the Cursors, you can save the scheme for future use by pressing the ***Save Preference Set*** button.

Sounds Page

On the **Sounds** page you can select a time and have HYSYS play a *.wav file if the Steady State solution takes longer than the selected time.

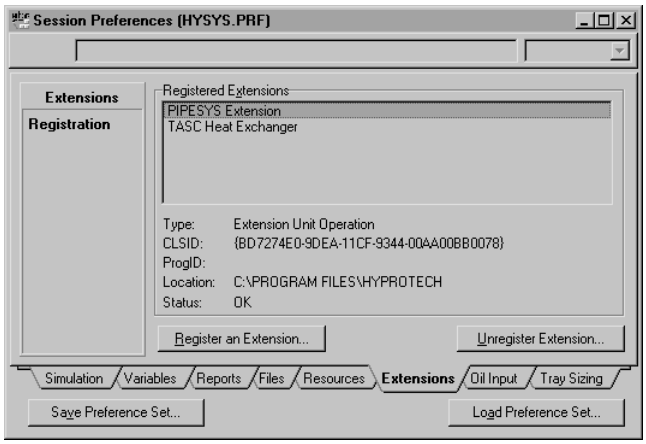
Figure 7.57



Extensions Tab

To register an extension, press the ***Register an Extension*** button. From the dialog, select the file you wish to register. To unregister an extension, select the extension from the **Registered Extensions** group and press the ***Unregister an Extension*** button.

Figure 7.58



Oil Input Tab

The pages on the **Oils** tab allows you to set default settings and select options for characterizing oils in your simulation case. This tab contains two pages, **Assay Definition** and **Assay Options**. You can specify the default options for defining an oil assay on these pages. For more details about these options, please see Chapter 3 - HYSYS Oil Manager in the **Simulation Basis** manual.

Tray Sizing Tab

From the **Tray Sizing** tab, you can set the HYSYS defaults for the **Tray Sizing Utility**. Any parameters set here will be automatically used when you attach a Tray Sizing utility to your case. For more information on this utility see Section 8.14 - Tray Sizing of this manual.

Parameters Page

From here you can set the **Auto Section Parameters** for the utility. These include the **Area Tolerance**, **NFP Diameter Factor** and the **Tray Internal Type**.

Packed Page

On this page, set the defaults for the **Packed Tray Setup Info.** This includes information on the **Correlation Type**, **Packing Flood Factor**, **Maximum Packing** and **Maximum Pressure Drop per Length**.

Trayed Page

On this page, select the type of tray using the radio buttons and then specify the geometry particular to that tray type.

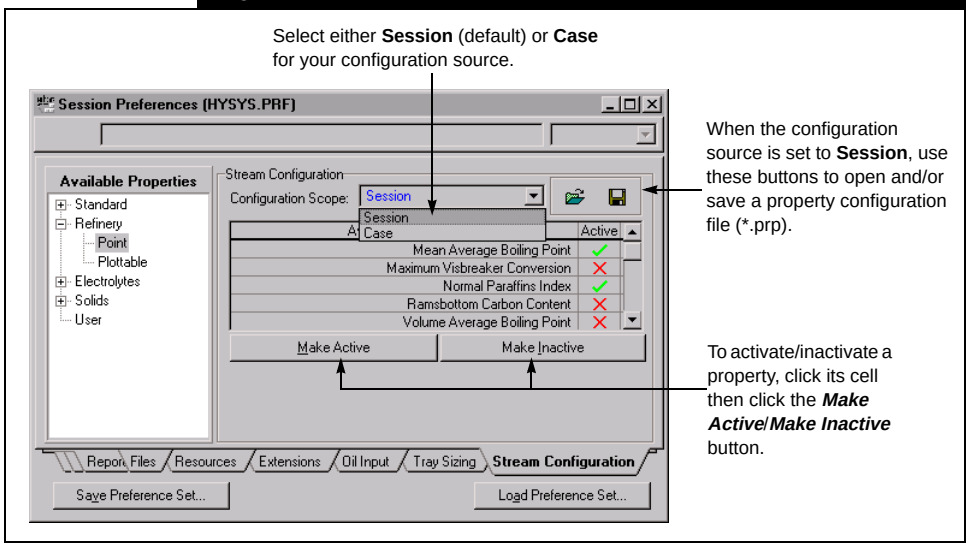
Stream Configuration Tab

This tab allows you to specify settings relating to the various Package Properties available. Package Properties in HYSYS are property groups with unique handling characteristics for dealing with specific component families, for example Electrolytes. Package Properties are accessible from any Stream property view where they can be viewed and plotted.

For details relating to the various Package Properties available with HYSYS, please contact your local Hyprotech agent.

The **Stream Configuration** tab contains a list of available properties. Only properties included with your version of HYSYS will be enabled in the view. Selecting a property group on the left of the view will allow you to configure **Point** and **Plottable** stream options relating to that property. A Point property shows you the property value calculated for the whole stream, while Plottable properties show how the property is distributed across the streams boiling range.

Figure 7.59



You can select how the stream configuration will be stored using the **Configuration Source** drop down list. The following options are available:

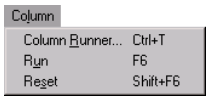
- **Session** - allows you to either save your configuration to an external property configuration file (*.prp) or open a previously saved configuration file for use during this session.
- **Case** - all stream configuration settings will be saved with the current case.

You can activate a property for use in the simulation by clicking the property's cell and then clicking the ***Make Active*** button. Activating a Package Property from the **Session Preferences** view sets the property preferences for the entire simulation case. The property becomes accessible to other objects within the case.

To activate a property for a specific stream only, do so on the flowsheet, from the stream's property view. For more details about these options, please see the Package Properties section in Chapter 2 - Streams of the **Steady State Modeling** manual.

7.8 Column

Figure 7.60



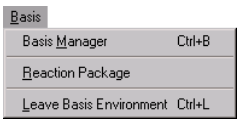
This menu item only appears inside the Column Environment. The options under Column are:

Command	Description
Column Runner	View the Column Runner.
Run	Starts the Column Solver.
Reset	Resets the Column Solver.

Refer to **Chapter 7 - Column** in the **Steady State Modeling** manual for more information.

7.9 Basis

Figure 7.61

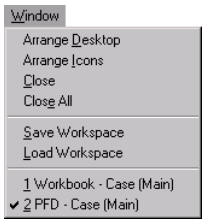


The Basis option appears when you enter the Basis Environment. The options available in this menu are:

Command	Description
Basis Manager	Opens the Simulation Basis Manager view. For more information, refer to **Section 7.3 - Simulation**.
Reaction Package	Accesses the Reaction Package view. Refer to **Section 7.4.1 - Reaction Package**, for more information on the functionality of this view.
Leave Basis Environment	Returns to the Simulation Building Environment (Main Environment).

7.10 Window

Figure 7.62



This menu contains general Windows application functions. The options are:

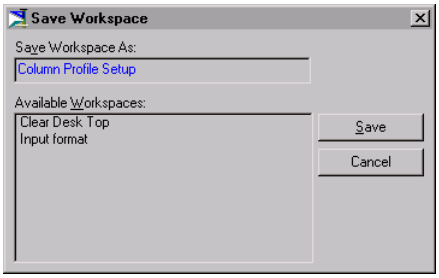
Command	Description
Arrange Desktop	Cascades all windows which are currently open and not iconized. Face plates will be placed in rows or column according to the specifications on the **Simulation** page of the **Session Preferences** view. Refer to **Section 7.7.5 - Preferences**, for more information.
Arrange Icons	Arranges icons horizontally at the bottom of the Desktop.
Close	Closes the active window.
Close All	Closes all windows.
Save Workspace	Saves the current window layout for future use.
Load Workspace	Loads another HYSYS case which is currently open. This function allows you to toggle between cases.

The last section in this menu lists all open windows on the DeskTop. The active window is indicated with a check mark.

7.10.1 Save Workspace

You can save different Workspace arrangements within a HYSYS case. The Workspace is a specific organization of views for the current case. For example, you could create an arrangement of views which has the PFD, Workbook, Controllers, Strip Charts, etc. You can name each arrangement individually, then re-access the arrangement at any time.

Figure 7.63

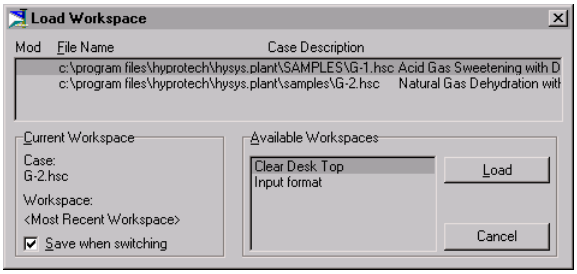


This has no effect on the calculation status. It is simply the way the various views are arranged. After changes have been made to the DeskTop arrangement, you can then re-load a saved arrangement to re-access the window layout.

7.10.2 Load Workspace

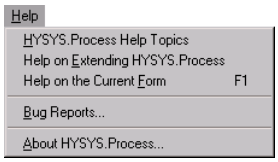
The **Load Workspace** dialog displays all cases that are currently open. The Workspaces contained within each case appear in the **Available Workspaces** group. When loading a Workspace, choose the case which contains the Workspace, select an **Available Workspace** and press the ***Load*** button.

Figure 7.64



7.11 Help

Figure 7.65



The options under **Help** are:

Command	Description
HYSYS Help Topics	Displays HELP contents.
Help on Extending HYSYS	Help topics for OLE functionality.
Help on the Current Form	Accesses the help topic for the active window.
Bug Reports	Accesses HYSYS' unique bug reporting option.
About HYSYS	Provides information about HYSYS.

8 Utilities

8.1 Boiling Point Curves 4
8.1.1 Generalized Procedure 4
8.1.2 Example Results 6
8.2 CO2 Solids 10
8.3 Cold Properties 12
8.4 Critical Property 14
8.4.1 Quick Start 14
8.4.2 True and Pseudo Critical Properties 15
8.5 Data Recon 16
8.6 Depressuring 17
8.6.1 Quick Start - Fire Wetted Example 18
8.6.2 Quick Start - Adiabatic Example 22
8.6.3 Design Tab 24
8.6.4 Tabular Results Tab 35
8.6.5 Graphical Results Tab 36
8.6.6 Notes Tab 37
8.6.7 Modelling Specific Systems 37
8.7 Derivative 40
8.8 Envelope 40
8.8.1 Connections 41
8.8.2 Plots 42
8.8.3 PV-PH-PS Envelopes 43
8.8.4 TV-TH-TS Envelopes 44

8.9 Hydrate Formation 45

8.9.1 Stream Settings 45
8.9.2 Hydrate Formation at Stream Conditions Group 47
8.9.3 Formation Temperature at Stream Pressure 49
8.9.4 Formation Pressure at Stream Temperature 49
8.9.5 Hydrate Inhibition 50
8.9.6 Hydrate Inhibition Example 51

8.10 Parametric Utility 52

8.10.1 Neural Networks 53
8.10.2 Variables 53
8.10.3 Generalized Procedure 54
8.10.4 PM Utility View 56

8.11 Pinch Utility 65

8.11.1 Quick Start 65

8.12 Pipe Sizing 69

8.12.1 Quick Start 69
8.12.2 Pipe Sizing View 71

8.13 Property Table 73

8.13.1 Quick Start 73

8.14 Tray Sizing 79

8.14.1 Quick Start 79
8.14.2 Tray Sizing — Design 81
8.14.3 Tray Sizing — Rating Part II 87

8.15 Tray Sizing Manager 89

8.15.1 Setup Page 90
8.15.2 Design Specifications 95
8.15.3 Specs Page 96
8.15.4 Tray Internals Page 101
8.15.5 Auto Section 106

8.16 User Properties 109

8.16.1 Quick Start 109

8.17 Vessel Sizing 114

8.17.1 Quick Start 114

8.18 Adjust Operation with Utility Target Object 118

8.19 References 120

The Utility commands are a set of tools which interact with a process by providing additional information or analysis of streams or operations. In HYSYS, Utilities become a *permanent* part of the Flowsheet and are calculated automatically when appropriate. They can also be used as target objects for **ADJUST** Operations.

The Utilities are accessed by selecting **Utilities** under **Tools** in the Menu Bar (you may also press CTRL U). Most utilities can also be added through the **Utilities** page found on the **Attachments** tab of a stream's property view. A utility added through either route is automatically updated in the other location. For example, if you attach an **Envelope** utility to a stream using the CTRL U route, it will automatically be displayed on the **Utilities** page of the **Attachments** tab in the property view of the stream to which it was attached.

You can select any of the following utilities from the **Available Utilities** view:

Utilities	Description
Boiling Point Curves	Obtain laboratory-style distillation results for streams.
CO2 Freeze Out	Determine stream CO2 freezing conditions.
Cold Properties	Calculate several stream Cold Properties (e.g. True and Reid Vapour Pressures, Flash Point, Pour Point, Refractive Index, etc.).
Critical Property	Calculate true and pseudo critical properties for streams.
Data Recon	The Data Recon utility is used by HYSYS.RTO optimization objects as a data holder that allows for multiple sets of stream data, each corresponding to a different set. For further details, refer to the HYSYS.RTO User's manual.
Depressuring	Model the pressure letdown of a single vessel or network of vessels under plant emergency conditions.
Derivative	The Derivative utility is used by HYSYS.RTO to hold all the data used for defining the RTO optimizer constraints and variables. For further details, refer to the HYSYS.RTO User's manual.
Envelope Utility	Show critical values and phase diagrams for a stream.
Hydrate Formation Utility	Determine stream hydrate formation conditions.
Parametric Utility	The Parametric Utility integrates Neural Network (NN) technology into its framework. The major function of the utility is to approximate an existing HYSYS model with a parametric model.
Pinch Utility	Optimize the use of process heat exchange and utilities for heat exchangers, LNG's, coolers and heaters.

Utilities	Description
Property Table	Examine stream property trends over a range of conditions.
Tray Sizing	Size/rate existing sections or full towers.
User Property	Define new stream properties based on composition.
Vessel Sizing	Size and cost installed Separator Unit Operations.

To add a Utility, highlight one of the available utilities (on the right side), and select the ***Add Utility*** button. You may view or delete an existing Utility simply by highlighting it (left side of view), and selecting the ***View Utility*** or ***Delete Utility*** button respectively. Select the ***Close*** button to close the **Available Utilities** view.

8.1 Boiling Point Curves

*Refer to Chapter 3 - HYSYS Oil Manager of the **Simulation Basis** guide for details on the distillation data types.*

The Boiling Point Curves utility, which will generally be used in conjunction with characterized oils from the Oil Manager, allows you to obtain the results of a laboratory style analysis for your simulation streams. Simulated distillation data including TBP, ASTM D86, D86 (Corr.), D1160(Vac), D1160(Atm) and D2887 as well as critical property data for each cut point and cold property data are calculated. The data can be viewed in tabular format or graphically.

*For a **tray section**, the boiling point curves and critical property data can be accessed on the **Profiles** tab of the Column Runner.*

The object for the analysis can be a stream, a phase on any stage of a tray section, or one of the phases in a separator, in a condenser or in a reboiler. You select the basis for the calculations and you can specify the boiling ranges for the simulated distillation data.

8.1.1 Generalized Procedure

*The list in the Object Navigator will be filtered according to your selection of **Object Type**.*

1. The Boiling Point Curves utility can be accessed by selecting **Utilities** under **Tools** in the Menu Bar. Choose **Boiling Point Curves** from the list in the **Available Utilities** view and press the ***Add Utility*** button.
2. On the **BP Curves** tab, change the **Name** of the utility, if desired.
3. Select the **Object Type**: Stream, Tray Section, Separator, Condenser, or Reboiler.
4. Press the ***Select Object*** button and choose the appropriate object from the Object Navigator. If the Object Type which you have selected is a Tray Section choose a **Stage** from the aptly named drop down menu.
5. Select a **Basis** for the calculation of the distillation data. The options are: Mole Frac, Mass Frac, Liquid Volume.

6. For all object types except the Stream selection, you can select the **Phase** for the analysis: Vapour or Liquid.
7. Press the ***Boiling Ranges*** button to access a property view from which you can specify the initial and final boiling point cut points on a liquid volume basis. You can also choose the D86 curve interconversion method from the **Default D86 Curve Type** drop down list:
   - **API (Edmister) Method**
   - **API 1988 (Riazi) Method**
   - **API 1992 modified (Riazi) D86**
   - **Edmister-Okamoto 1959 Interconversion**
8. View the results of the boiling point curve calculations in tabular format on the **Performance** tab, **Results** page. Simulated distillation profiles are provided for the following assay types: TBP, ASTM D86, D86 Corr., ASTM D1160 (Vac.), ASTM D1160 (Atm.), ASTM D2887.

*Refer to **Section 3.6 - Oil Characterization View** of the **Simulation Basis** manual for more information on the D86 interconversion options.*

9. Switch to the **Critical Properties** page where you can examine a table containing, for each cut point, the Critical Temperature, Critical Pressure, Acentric Factor, Molecular Weight and Liquid Density.
10. On the **Cold Props** page, you will find a table of various cold properties and the calculated ratio of paraffins, naphthas and aromatics.

*Refer to **Section 8.3 - Cold Properties** for more information on the Cold Properties.*

11. The **Plots** tab shows the Boiling Point Curves results and the Critical Properties results in graphical form. Examine the plot of your choice by making a selection from the **Dependent Variable** drop down list:
    - Boiling Point Curves
    - Critical Temperature
    - Critical Pressure
    - Acentric Factor
    - Molecular Weight
    - Liquid Density
12. You can customize a plot by object inspecting the plot area and selecting **Graph Control** from the menu.

*The ASTM D86 boiling point curve corresponds to the **true** boiling points of the oil, which assumes no cracking has occurred.*

*When the oil is characterized by a ASTM D86 distillation assay **with no** cracking option, the D86 Corr boiling point curve corresponds to the assay input data. The ASTM D86 boiling point curve then corresponds to raw lab data, with no cracking correction applied.*

*When the oil is characterized by a ASTM D86 distillation assay **with** cracking option, the ASTM D86 boiling point curve corresponds to the assay input data. The cracking correction factor is then applied to the D86 Corr boiling point curve.*

*Refer to **Section 6.4 - Graph Control** of the **User's Guide** for details concerning the customization of plots.*

8.1.2 Example Results

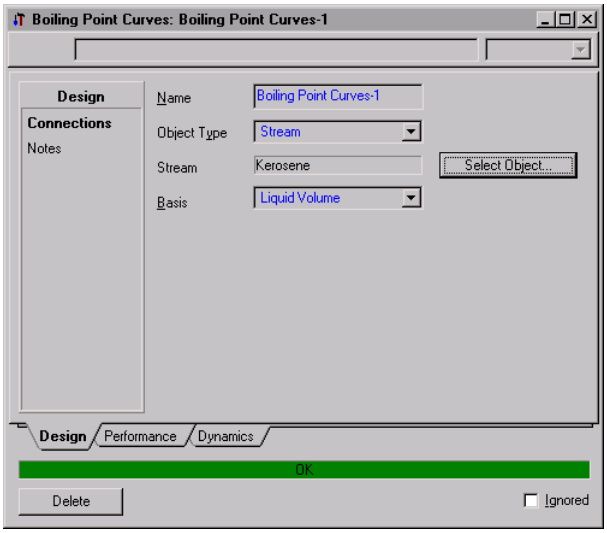
If you would like to proceed through an example and obtain the following results from the Boiling Point Curves utility, open case: **R-1.hsc**, which can be found in the...*HYSYS**Samples* directory**.**

In the Oil Manager, assay data is used to characterize the raw crude feed stream. The crude feed is processed into five products in a 29 stage crude fractionator which has three attached side strippers.

Simulated distillation data, critical properties and cold properties will be examined for the **Kerosene** product stream.

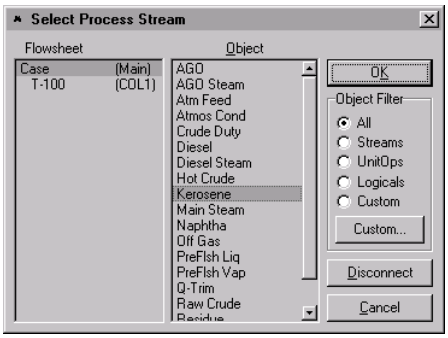
Setting the Utility Parameters

Figure 8.1



On the **Boiling Point Curves** tab, you select the parameters for the Boiling Point Curves utility. Ensure the **Basis** is set to **Liquid Volume** and leave the **Object Type** at its default, Stream. Press the ***Select Object*** button to access the Object Navigator. The selection of the **Object Type** filters the list shown in the navigator. Select the stream **Kerosene** as shown.

Figure 8.2



Since a stream was selected as the Object Type, you do not need to specify a phase for the calculations. HYSYS will calculate the distillation data for the bulk phase of the selected stream and display the information in the **Results** table. Examine the range for the simulated distillation data profiles by pressing the ***Boiling Ranges*** button. The initial and final boiling point defaults will be used for this example.

Critical Properties

You can switch to the **Critical Properties** tab to examine the critical properties for each cut point. A section of the results table is shown here.

Figure 8.3

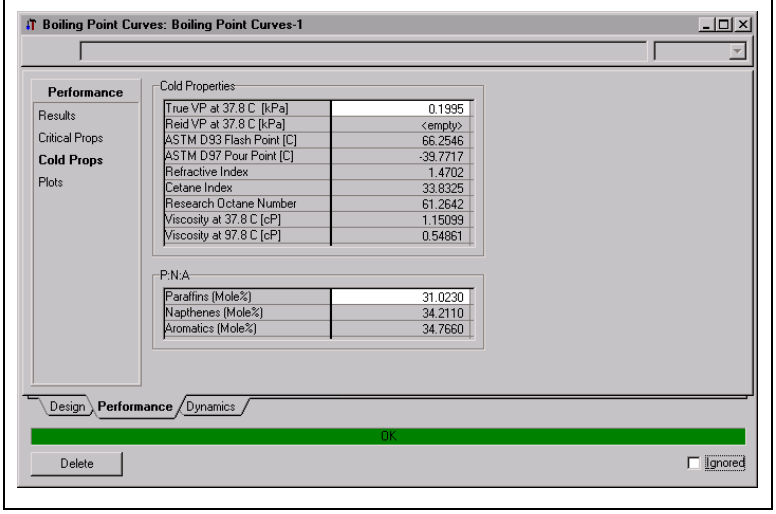
Cut Point [%]	Critical Temp [C]	Critical Press [kPa]	Acentric Factor	Mole Wt.	Liquid Density [kg/m3]
0.00	172.6	3824	0.1224	58.290	620.0
1.00	340.7	2917	0.3860	120.325	797.6
2.00	346.2	2867	0.3957	123.423	804.6
3.50	355.9	2791	0.4113	128.567	810.9
5.00	361.5	2748	0.4205	131.584	814.4
7.50	365.3	2717	0.4272	133.826	816.8
10.00	369.2	2687	0.4339	136.069	819.1
12.50	373.0	2657	0.4406	138.312	821.4
15.00	376.9	2628	0.4473	140.555	823.7
17.50	379.7	2603	0.4532	142.530	825.4
20.00	382.9	2582	0.4580	144.170	826.5
25.00	387.5	2546	0.4666	147.065	829.9
30.00	391.3	2516	0.4739	149.561	832.1
35.00	394.6	2490	0.4803	151.791	833.8

Cold Properties

Details of the methods used to determine the Cold Properties can be found in Section 8.3 - Cold Properties.

You can view the bulk cold properties of the stream on the **Cold Properties** page. Also listed is the ratio of paraffins to naphthas to aromatics.

Figure 8.4



Plots

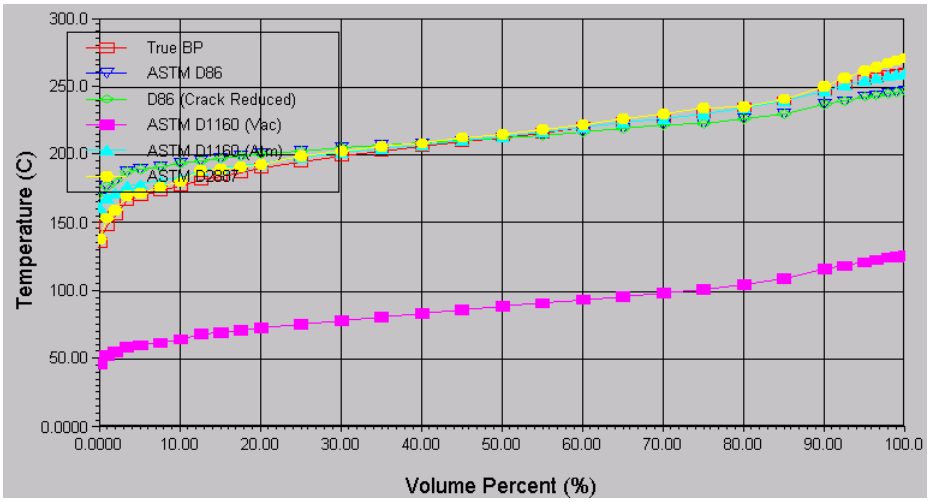
On the **Plots** page, graphical profiles of either the simulated distillation data or the critical properties can be examined. The profiles of the TBP and D86 curves will be examined.

*See Section 6.4 - Graph Control of the **User's Guide** for full details on the options available from the **Graph Control** property view.*

To view only the TBP and D86 profiles, modifications must be made to the plot. The default selection for the **Dependent Variable**, which is **Boiling Point Curves**, can remain. When the plot first appears, all of the simulated distillation data profiles are present. The unwanted profiles can be eliminated via the **Graph Control** property view. Access this view by object inspecting the plot area and choosing **Graph Control**.

By changing the y-axis range and the format of the x-axis values, the plot of the simulated distillation data profiles will be as shown.

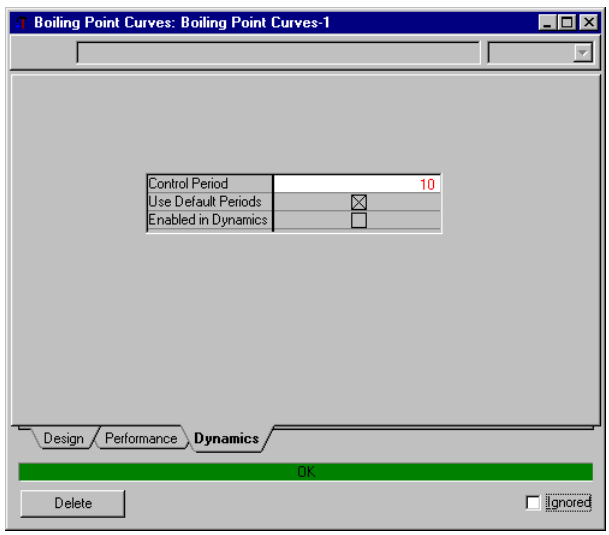
Figure 8.5



Dynamics

The **Dynamics** tab allows you to control how often the utility gets calculated when running in Dynamic mode.

Figure 8.6



The Control Period field is used to specify the frequency that the utility is calculated. A value of 10 indicates that the utility will be recalculated every 10th pressure flow step. This can help speed up your dynamic simulation since utilities can require some time to calculate.

The Use Default Periods check-box allows you to set the Control Period of one utility to equal the Control Period of any other utilities that you have in the simulation. For example, if you have five utilities and require them all to have a Control Period of 5 and currently the value is 8, with this check-box activated if you change the value in one utility all the other utilities will change. Alternatively if you want all the utilities to have different values you would de-active this check-box.

The Enable in Dynamics check-box is used to activate this feature for use in Dynamic mode.

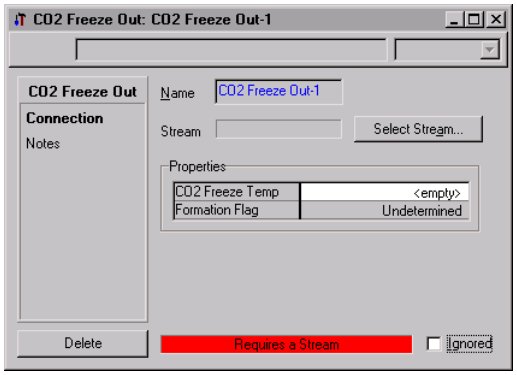
8.2 CO2 Solids

An equation-of-state based approach is used to calculate the incipient solid formation point for mixtures containing Carbon Dioxide (CO2). The model can be used for predicting the initial solid formation point in equilibrium with either vapours or liquids. The fugacity of the resultant solid is obtained from the known vapour pressure of solid CO2. The fugacity of the corresponding phase (in equilibrium with the solid) is calculated from the equation of state.

> CO2 Solids prediction is restricted to the Peng Robinson (PR) and Soave Redlich Kwong (SRK) equations of state.

You are required to enter the stream for which the calculations will be made. The stream is chosen from the Object Navigator, which is accessed by pressing the **Select Stream** button.

Figure 8.7



HYSYS will determine the CO_2 Freeze Temperature, and will display the formation status in the **Formation Flag** cell:

Formation Flag	Flag Significance
Undetermined	No Stream has been chosen.
NO CO2 in Stream	There is no CO_2 present in the Stream.
Does NOT Form	Solid CO_2 will not form at the present conditions of the stream. The CO_2 Freeze Temperature will be shown in the corresponding field.
Solid CO2 Present	Solid CO_2 is present at the current stream conditions. The CO_2 Freeze Temperature is shown in the corresponding field.

An equation-of-state based approach is used to calculate the incipient solid formation point for mixtures containing Carbon Dioxide (CO2). The model can be used for predicting the initial solid formation point in equilibrium with either vapours or liquids. The fugacity of the resultant solid is obtained from the known vapour pressure of solid CO2. The fugacity of the corresponding phase (in equilibrium with the solid) is calculated from the equation of state.

CO2 Solids prediction is restricted to the Peng Robinson (PR) and Soave Redlich Kwong (SRK) equations of state.

To ignore this Utility during calculations, select the **Ignored** check box. HYSYS will disregard the utility entirely until you restore it to an active state by clearing the check box.

8.3 Cold Properties

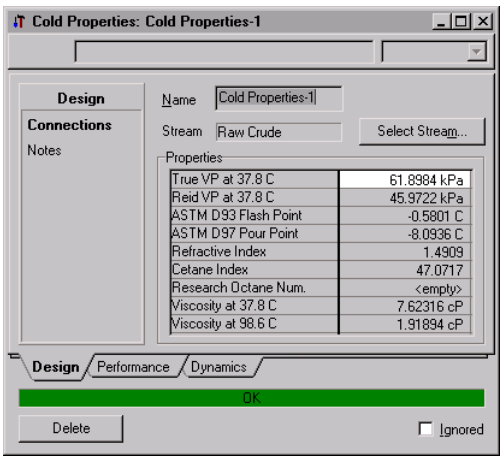
The following list summarizes the cold properties which are available through the **Cold Properties** utility:

Cold Property	Calculations	Range of Validity
True Vapour Pressure @ 100°F (37.8°C)	Vapour Pressure method of selected property package	P>1.5 kPa
Reid Vapour Pressure @ 100°F (37.8°C)	Vapour pressure of system when vapour:liquid ratio by volume is 4:1	P>1.5 kPa
Flash Point	As per API 2B7.1	150°F<ASTM D86 10% (or NBP)<1150°F, -15°F<Flash Point<325°F
Pour Point	As per API 2B8.1	140<MW<800, 1<API gravity<50, -110°F<Flash Point<140°F
Refractive Index	As per API 2B5.1-1	70<MW<600, 97°F<NBP<1000°F, 0.63<sg<1.1, 1.35<Refractive Index at 20°C<1.65
Cetane Index (Diesel Index)	Proprietary method	300°F<D86 10%<700°F
Research Octane Number (R.O.N.)	Proprietary method	D86 50% ~420°F
Viscosity at 100°F (37.8°C)	See the **Viscosity** section in **Appendix A - Property Methods and Calculations** in **Simulation Basis** manual.	
Viscosity at 210°F (98.6°C)	See the **Viscosity** section in **Appendix A - Property Methods and Calculations** in Simulation Basis guide.	
ASTM D86 Distillation Curve	API Figure 3A1.1 (1963)	51°F<TBP 10%<561°F
P/N/A (mol%)	As per API 2B4.1	MW>70

To add the Cold Property Utility, select **Utilities** under **Tools** in the Menu Bar, highlight **Cold Properties**, and press the ***Add Utility*** button.

Attach a stream to the utility from the Object Navigator, accessed via the ***Select Stream*** button.

Figure 8.8



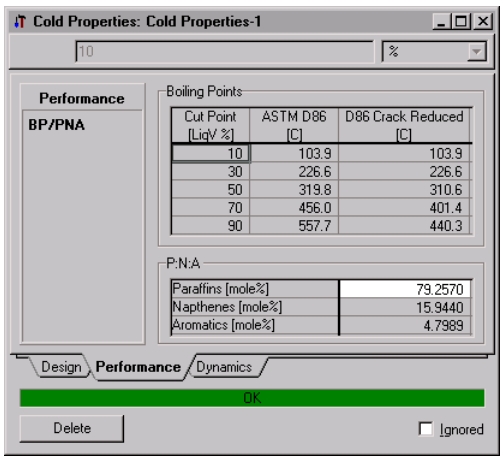
The **Notes** page can be used to write notes and it has the same functionality as of a Notepad.

The **Properties** tab displays the following properties:

- True Vapour Pressure
- Reid Vapour Pressure
- Flash Point
- Pour Point
- Refractive Index
- Cetane Index
- Research Octane Number
- Viscosity at 100°F (37.8°C) and 210°F (98.6°C)

The **BP/PNA** tab displays the ASTM Distillation Curve (ASTM D86 10%, 30%, 50%, 70%, 90% Points), and the P/N/A mole percents.

Figure 8.9



The **Dynamics** tab allows you to control how often the utility gets calculated when running in Dynamic mode (refer to the Dynamics sub-heading in **Section 8.1 - Boiling Point Curves** for more information).

To ignore this Utility during calculations, select the **Ignored** check box. HYSYS will disregard the utility entirely until you restore it to an active state by clearing the check box.

8.4 Critical Property

The Critical Property Utility calculates both the true and pseudo critical temperature, pressure, volume and compressibility factor for a fully defined stream.

8.4.1 Quick Start

Before the Critical Property Utility is accessed, set up a Fluid Package using the **Peng Robinson** property method and the required components. Install the stream **Feed**.

MATERIAL STREAM [Feed]		
Tab [Page]	**Input Area**	**Entry**
Worksheet [Conditions]	Temperature	60.0000 °F
	Pressure	600.0000 psi
	Molar Flow	100.0000 lbmole/hr
Workhsheet [Composition]	Nitrogen Mole Frac	0.0068
	CO2 Mole Frac	0.0138
	C1 Mole Frac	0.4827
	C2 Mole Frac	0.1379
	C3 Mole Frac	0.0690
	i-C4 Mole Frac	0.0621
	n-C4 Mole Frac	0.0552
	i-C5 Mole Frac	0.0483
	n-C5 Mole Frac	0.0414
	n-C6 Mole Frac	0.0345
	n-C7 Mole Frac	0.0276
	n-C8 Mole Frac	0.0207

Access the **Critical Property** utility by selecting **Utilities** under **Tools** in the Menu Bar. Choose **Critical Properties** from the list of utilities and press the ***Add Utility*** button.

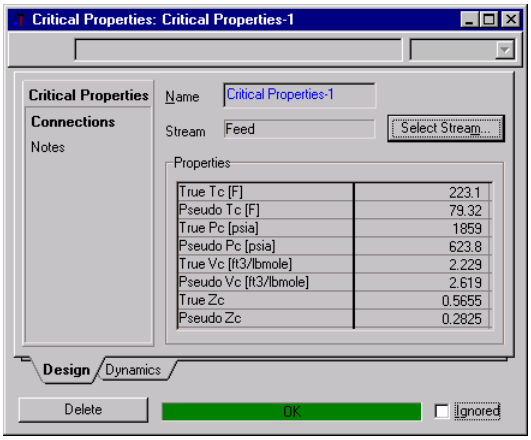
The general procedure for the Critical Property Utility is as follows:

1. On the **Critical Property** view, change the name of the utility, if desired.
2. Press the ***Stream*** button.
3. Select the stream **Feed** from the Object Navigator.
4. View the critical property analysis for the selected stream in the **Calculated Property Values** group box.

Critical Property Analysis

Select Feed from the Object Navigator and examine the critical property values for the stream.

Figure 8.10



The **Dynamics** tab allows you to control how often the utility gets calculated when running in Dynamic mode (refer to the Dynamics sub-heading in Section 8.1 - Boiling Point Curves for more information).

8.4.2 True and Pseudo Critical Properties

The Critical Properties utility displays two sets of critical properties, true and pseudo critical properties. **True Critical Properties** are those properties calculated using the mixing rules associated with the property package chosen. **Pseudo Critical Properties** use simple linear models to estimate the critical properties of a mixture. They are often very different from the true critical points and **have no real physical significance**, but sometimes are used in empirical correlations.

Mathematically, the pseudo critical temperature, pressure and compressibility (T_{pc}, P_{pc} and Z_{pc}) are defined as:

$$T_{pc} = \sum_{i=1}^{n} y_i T_{ci} \qquad (8.1)$$

$$P_{pc} = \sum_{i=1}^{n} y_i P_{ci} \tag{8.2}$$

$$Z_{pc} = \sum_{i=1}^{n} y_i Z_{ci} \tag{8.3}$$

where: y_i *= mole fraction of component i.*

n *= total number of components in mixture*

T_{ci} *= the critical temperature of component i*

P_{ci} *= the critical pressure of component i*

Z_{ci} *= the critical compressibility of component i*

The remaining pseudo critical property, pseudo critical volume v_{pc}, is calculated using the following relationship:

$$v_{pc} = \frac{Z_{pc} T_{pc} R}{P_{pc}} \tag{8.4}$$

8.5 Data Recon

The **Data Recon** utility is a component of the HYSYS.RTO real-time optimization package available as a plug-in to the basic HYSYS software package. The **Data Recon** utility is one of two utilities used by HYSYS.RTO to provide the primary interface between the flowsheet model and the solver. Their primary purpose is to collect appropriate optimization objects which are then exposed to solvers to meet a defined solution criteria.

Please refer to the HYSYS.RTO User's manual for details concerning the use of this utility. This manual details all features and components related to the HYSYS real time optimization package.

If your current HYSYS version does not support RTO, contact your local Hyprotech representative for more details.

8.6 Depressuring

The **Depressuring** utility allows you to examine temperature and pressure profiles during simulated gas blowdowns and pressure letdowns of vessels under plant emergency conditions. The contents of vessels are represented by streams of known composition. You can specify liquid and vessel volumes or have HYSYS calculate these values based on the molar flowrate and liquid fraction of the stream in question. The three operation modes available are fire, adiabatic and isothermal. You can examine the results of the depressuring calculations in tabular or graphical format.

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

There are four types of depressuring calculation available:

Calculation	Description
Fire Mode	Used to simulate plant emergency conditions that could occur during a plant fire. Pressure, temperature and flow profiles are calculated for the application of an external heat source to a vessel, piping or combination of items. Heat flux into the fluid is user defined. You must not have a wetted area specified for this calculation
Wetted Fire Mode	As above except that the Heat flux into the fluid is calculated from the API equations for a fire to a liquid containing vessel. A wetted area for the vessel is required. This wetted area is the area used for heat transfer in the model
Adiabatic Mode	Used to model the gas blowdown of pressure vessels or piping. No external heat is applied. Heat flux between the vessel wall and the fluid is modelled as the fluid temperature drops due to the depressurisation. The heat transfer coefficient can be input by the user or can be calculated by HYSYS from the vessel fluid’s vapour properties. When it is estimated by HYSYS, the heat transfer coefficient is estimated from the “wetted” area and the vessel volume input by the user. The “wetted area” supplied should be equal to the total surface area of the vessel, not the area in contact with the liquid. Typical use of this mode is the depressuring of compressor loops on emergency shutdown.
Isothermal Mode	The temperature of the vessel fluid remains constant. HYSYS determines the necessary heat flux required achieving isothermal depressuring.

To illustrate the versatility of the depressuring utility, two examples will be shown. The first will demonstrate the use of Fire mode for a network of vessels and the second will show the depressuring utility in Adiabatic mode. Both example sessions use a common initial setup, including the fluid package and installed streams.

8.6.1 Quick Start - Fire Wetted Example

The following example will illustrate the use of the Depressuring utility in Fire Mode for a network of vessels. The primary use of the fire mode analysis is to simulate plant emergency conditions. To model the events which would occur during a plant fire, HYSYS calculates the time dependent pressure and temperature profiles of a fluid during the application of an external heat source to a vessel or to a group of vessels.

Initial Setup

First, install a Fluid Package using the **Peng Robinson** property method and the required components. The network of two vessels will be represented by the streams **Tank1** and **Tank2**. Install the streams as shown. Note that the flows have been set to 1 lbmole/hr however these values are not necessary for the calculation.

MATERIAL STREAM [Tank1]		
Tab [Page]	**Input Area**	**Entry**
Worksheet [Conditions]	Temperature	75.0000 °F
	Pressure	600.0000 psi
	Molar Flow	1.0000 lbmole/hr
Worksheet [Composition]	Methane Mole Frac	0.5600
	Ethane Mole Frac	0.2000
	Propane Mole Frac	0.1000
	i-Butane Mole Frac	0.0500
	n-Butane Mole Frac	0.0500
	i-Pentane Mole Frac	0.0200
	n-Pentane Mole Frac	0.0200

MATERIAL STREAM [Tank2]		
Tab [Page]	**Input Area**	**Entry**
Worksheet [Conditions]	Temperature	75.00 °F
	Pressure	600.0 psi
	Molar Flow	1.000 lbmole/hr
Worksheet [Composition]	Methane Mole Frac	0.17792
	Ethane Mole Frac	0.19832
	Propane Mole Frac	0.19252
	i-Butane Mole Frac	0.13511
	n-Butane Mole Frac	0.14921
	i-Pentane Mole Frac	0.07191
	n-Pentane Mole Frac	0.07501

Access the Depressuring utility by choosing **Utilities** under **Tools** in the Menu Bar. Highlight **Depressuring** and press the ***Add Utility*** button.

If the vessel and liquid volumes are not specified, HYSYS will calculate the vessel volume based on a one hour liquid volume (liqvol) flowrate and the liquid volume as the vessel volume multiplied by the liquid fraction of the stream.

Attach Stream button

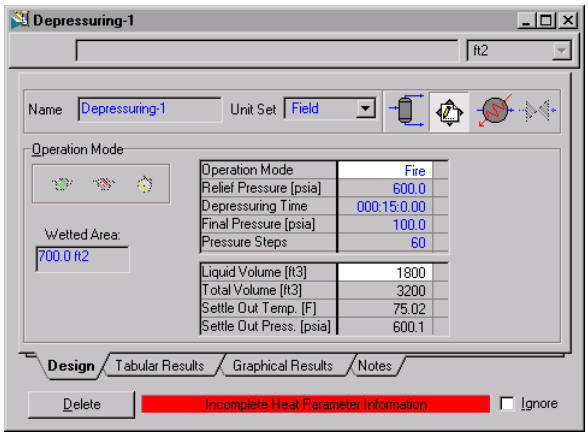
Generalized Procedure

1. On the **Design** tab, change the name of the utility, if desired.
2. Change the **Unit Set** to Field.
3. In the **Vessel Parameters** group box, click the ***Attach Stream*** button which opens the Object Navigator. Select streams **Tank 1** from the **Object** list box and press the ***OK*** button. Select the an empty cell and repeat the same procedure to add stream **Tank 2.**
4. Enter the **Vessel Volume** for both streams as **1600 ft^3**.
5. Specify **Liquid Volumes** of **500 ft^3** and **1300 ft^3** for **Tank1** and **Tank2**, respectively.

Operation Mode Setup button

6. Move to the **Operation Mode Setup** page. Since we have a **Wetted Area** of **200 ft^2** and **500 ft^2** for **Tank 1** and **Tank2,** respectively, enter a value of **700 ft2** for the **Wetted Area**. The view should appear similar to **Figure 8.11**

Figure 8.11



7. Examine the combined vessel liquid volume, total volume, wetted area, settle out temperature and settle out pressure.

__Fire Wetted__ mode, which is available only if a wetted area has been supplied, replaces __Fire__ mode in this situation.

8. The mode of operation is selected on the **Operation Mode Setup** page. For this example, **Fire** mode is required, but since wetted areas were supplied previously, **Fire Wetted** mode will automatically be used by HYSYS.

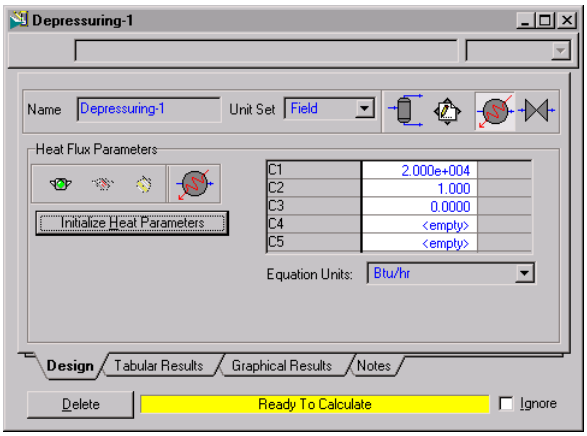
To simulate a pressure build-up, specify a relief pressure greater than the settle out pressure.

9. To simulate a pressure increase, specify a **Relief Pressure** of **630 psia**, which is above the combined vessel settle out pressure. The relief valve will not open until the vessel pressure has equalled or surpassed the relief pressure. The **Final Pressure** is set to **300 psia**. Set **Pressure Steps** parameter to **60**.

Heat Flux Parameter Setup button

10. Switch to the **Heat Flux Parameter Setup** page of the **Design** tab to view the equation and the parameters of the externally applied heat flux. Click ***Initialize Heat Parameters***. The **Heat Flux Parameter Setup** page, as shown in Figure 8.12, is produced.

Figure 8.12



*Press the **Equation Help** button on the **Heat Flux** or **Valve Eqns** tab for details on the available equations.*

11. Switch to the **Valve Parameters Setup** page.
12. Select the **Masoneilan** option from the **Equation** drop down list. In the **Av or Cv** field enter the value **50**.
13. Press the ***Green Light*** button to calculate.

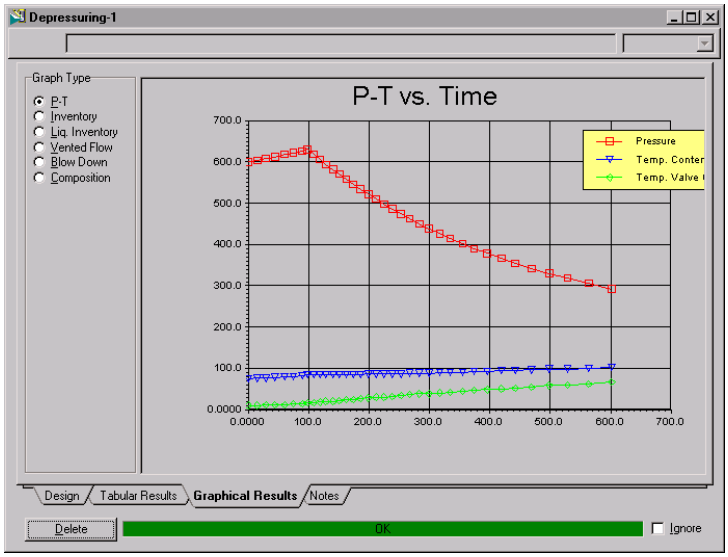
Valve Parameters Setup button

Depressuring Results

14. Switch to the **Graphical Results** tab to examine graphical representations of time dependent plots for the depressuring calculations. Select a radio button to determine the y-axis variable: **P-T** (Pressure - Temperature), **Inventory**, **Liq. Inventory**, **Vented Flow**, **Blow Down** or **Composition**.

*To edit a plot, object inspect the plot area and select **Graph Control** from the menu. For details on manipulating plots, refer to **Section 6.4 - Graph Control**.*

Figure 8.13



15. On the **Table Results** tab, you can examine the results of the depressuring calculations in tabular format.

8.6.2 Quick Start - Adiabatic Example

The following example will illustrate the use of **Adiabatic** mode for the Depressuring Analysis. The Adiabatic operation mode can model the gas blowdowns of pressure vessels. There is no external heat being applied to the vessel in this mode. However, there is a heat flux created between the vessel wall and the fluid as the fluid temperature drops because of the pressure reduction.

For this example you will require the stream **Tank 2** specified in **Section 8.6.1 - Quick Start - Fire Wetted Example**.

Adiabatic Input

The specifications and modifications to the defaults, as shown below, are required for the adiabatic mode example. Complete each tab with the information shown.

ADIABATIC INPUT MODIFICATIONS		
Tab [Page]	**Input Area**	**Entry**
Design [Vessel Parameters]	Stream	Tank2
	Vessel Volume	1600 ft3
	Liquid Volume	1300 ft3
	Wetted Area	500 ft2
Design [Operation Mode]	Mode	Adiabatic
	Relief Pressure	600 psi
	Final Pressure	300 psi
Design [Heat Flux Parameters]	Isentropic Efficiency	25%
	Vess. Material Specific Heat	0.12 Btu/lbF
	Vessel Mass	13000 lb
Design [Valve Parameters]	Valve Equation	Masoneilan
	Cv	50

Adiabatic mode requires the specification of an isentropic efficiency in percentage. The resulting range of values can be anywhere between zero, which represents an *isenthalpic process*, and one hundred percent, which represents an *isentropic expansion*.

*The use of the tern **Fluid** in the depressuring utility refers to a composite mixture of vapour and liquid phases. The **Heat Transfer Coefficient** value used on the **Heat Flux Parameters** page should reflect the relative volumes of each phase when estimating.*

Although there is no external source of heat applied to the vessel, heat is transferred from the vessel wall to the fluid. The variables used in the heat transfer during the depressuring are specified. The heat transfer area corresponds to the supplied wetted area and cannot be modified from the **Heat Flux** page. HYSYS calculates heat transfer to the liquid only and neglects the heat transfer effects to the vapour.

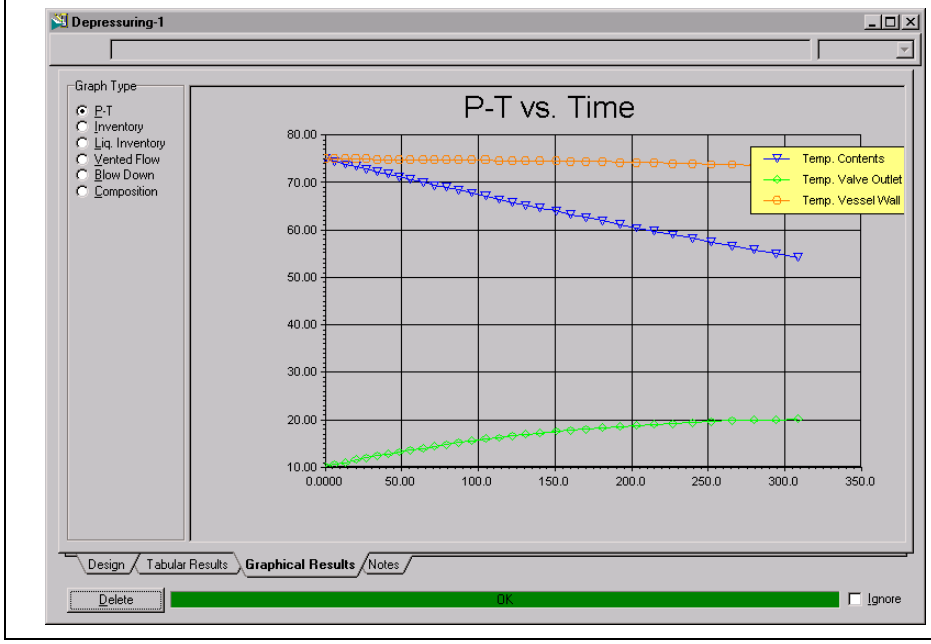
Press the ***Green Light*** button to run the depressuring utility.

To edit a plot, object inspect the plot area and select ***Graph Control*** *from the menu. For details on manipulating plots, refer to* ***Section 6.4 - Graph Control****.*

Results

On the **Graphical Plots** tab, choose the **P-T** radio button to examine the temperature profiles of the fluid, the vessel wall and the valve. You will need to change the y-axis and remove the pressure plot.

Figure 8.14



8.6.3 Design Tab

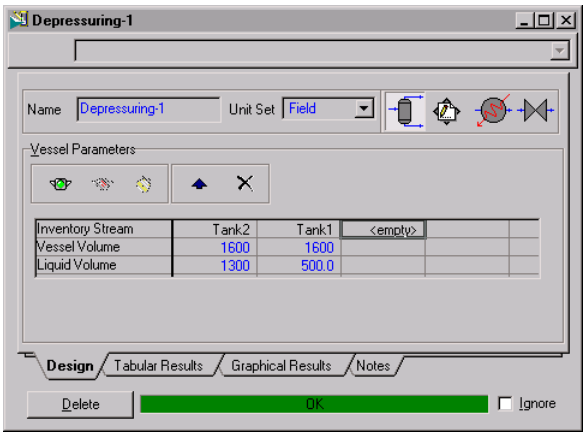
Figure 8.15

The traffic lights displayed on the Design tab are used to control the utility. There are three lights.

- Green - Runs the Depressuring utility
- Red - Stops the Depressuring utility
- Yellow - Restarts the utility after is has been stopped

Vessel Parameters Setup Page

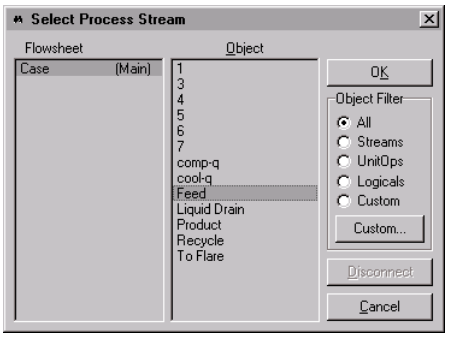
Figure 8.16



Attach Stream button

On the **Vessel Parameters Setup** page you must choose the stream(s) that represent the fluid you wish to depressure. This is done by clicking the ***Attach Stream*** button and selecting one of the available streams from the Object Navigator as shown in Figure 8.17.

Figure 8.17



You may also remove or change the streams to be used by clicking either the ***Remove*** or ***Edit*** buttons.

The following fields are available for each stream chosen:

Vessel\Liquid Volume Fields

For each stream selected for depressuring, HYSYS requires the Vessel Volume and the normal expected Liquid Volume of the vessel i.e.: at the normal liquid level. If the feed stream is two phase the composition of the liquid will be calculated from this.

You have three options for specifying these fields:

- If you do not provide values for the Vessel Volume or Liquid Volume, HYSYS will calculate values based on a 1 hour residence time. The resulting values will be:

$$Vessel\ Volume = \frac{Mass\ Flow}{Mass\ Density} \times 1\ hour \tag{8.5}$$

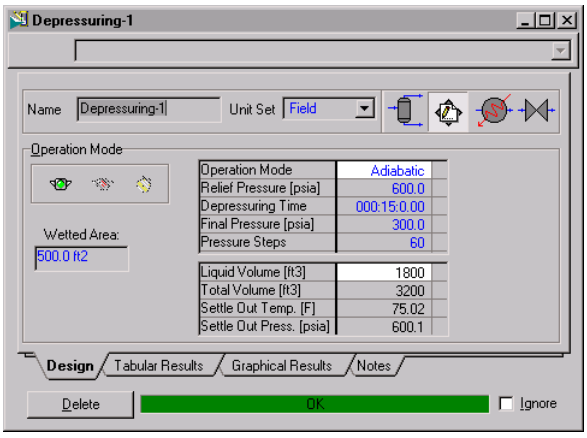
$$Liquid\ Volume = Liquid\ Volume\ Flow\ of\ Liquid\ Phase \times 1\ hour \tag{8.6}$$

- If you specify only the Vessel Volume, HYSYS estimates the Liquid Volume using a relationship similar to **Equation (8.6)** except the "*1 hour*" term is replaced by the vessels new residence time (i.e. Vessel Volume/Volumetric Flow). The remainder of the vessel is assumed to be filled with equilibrium vapour.
- If you specify both Liquid and Vessel volumes then the head space will be filled with equilibrium vapour.

Operation Mode Setup Page

On this page you must specify how the depressuring calculation will be done. This page consists of six fields that require specification.

Figure 8.18



Mode

Fire Wetted mode is automatically used by HYSYS when in Fire mode and a wetted area is specified.

From this drop down field you must select the required depressuring mode. The available choices are: **Fire Mode**, **Adiabatic Mode** and **Isothermal Mode**. (See **Section 8.6 - Depressuring Introduction** for a description of the 4 modes).

Relief Pressure

This field's value will depend on the mode chosen:

Depressuring Mode	Relief Pressure Value
Adiabatic type depressurisation	The relief pressure is set to the settle out pressure of the sources.
Fire relief	Could be equal to the design pressure of the vessel or piping; i.e. the set pressure of the relief valves on the system. If the relief pressure in Fire mode is higher than the settleout pressure then the heat flux will initially raise the system pressure up to the relief pressure, thus determining the time for the relief valve to actually lift.

Depressuring Time

The **Depressuring Time** is the time you wish this operation to take. It is defaulted as 15 minutes (900 sec) based on API 521, but you can alter this if required.

Final Pressure

Final Pressure is defaulted as atmospheric. Based on API it is normal to depressure to 50% of the starting pressure or to 100 psig (6.89 barg), however, if the depressuring time is reached (for API 521, 15 minutes) before the **Final Pressure** achieved calculations will stop. You may alter the **Final Pressure** value if required.

Pressure Steps

HYSYS will stop the depressuring calculations when either the *final pressure*, the *depressuring time* or the *number of calculation steps* is reached. For simple systems, 30 pressure steps should be adequate; if not, increase this number.

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

Wetted Area

Wetted area is the area of the vessel that is used to calculate the heat flux to the fluid. The way in which this is used depends on the mode chosen in the **Mode** field. This value represents the total wetted area of all of the specified vessels.

Calculation Mode	Area Usage
Fire/Wetted Fire Mode	Heat flux into the fluid is calculated from the API equations for a fire to a liquid containing vessel. The wetted area is the heat transfer area.
Adiabatic	Heat flux between the fluid and the vessel wall is calculated from the user-specified area and the temp difference. Heat transfer coefficient is either user specified at a fixed value or if left blank HYSYS will calculate it at each time step. The user specified area is the “wetted area” entered by the user. The heat transfer coefficient is specified or calculated from vapour properties. If the heat transfer coefficient is calculated by HYSYS, the calculation methods used are based on heat transfer from the entire vessel, and the user should enter a “wetted area” equivalent to the total surface area.

Settle Out Temperature and Pressure

Once the individual vessels are specified HYSYS initiates a Volume/Enthalpy analysis (flash) on the mixture. The analysis determines the required pressure and the temperature such that **Total Vessel** and **Liquid Volumes** of the mixture equal the sum of the Volumes for the individual vessels, as well as matching the mixture Enthalpy to the sum of the individual Enthalpies. The calculated Temperature and Pressure for this mixture are termed the *Settle Out Temperature* and *Pressure*.

In the case of an Individual vessel, the reported *Settle Out Temperature* and *Pressure* will be the system conditions. The exception is in the case where you specify the Liquid Volumes equal to the *Total Vessel Volume*, the *Settle Out Pressure* which will be reported is the pressure necessary to produce some vapour at the current temperature.

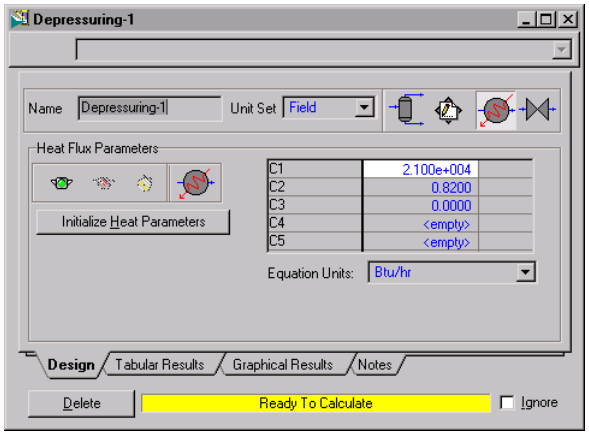
Heat Flux Parameter Setup Page

*Press the **Equation Help** button on for details on the available equations.*

The information required by the **Heat Flux** page of the **Design** tab will depend on the mode chosen in the **Operation Mode** drop down box found on the **Operation Mode Setup** page.

Fire Mode

Figure 8.19



When depressuring in **Fire** mode, five coefficients C1 to C5 are required specifications in order to set up the following generalised equation:

$$Q = C_1 + C_2 t + C_3(C_4 - T) + C_5\left(\frac{V_t}{V_0}\right) \qquad (8.7)$$

As an example, you could model the standard heat transfer equation:

$$Q = UA\Delta T \qquad (8.8)$$

By setting C_1, C_2 and C_5 to zero. Set C_3 to UA and C_4 to the constant temperature in the D*T* term.

The **Heat Flux Parameters** page for depressuring in **Wetted Fire** mode is very similar to the view observed in **Fire** mode. Three coefficients C1 to C3 need to be specified in order to set up the following equation, which is an extension to the standard API equation for flux to a liquid-containing vessel.

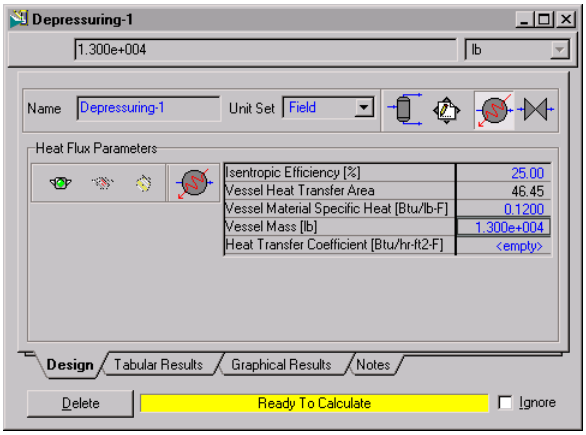
$$Q = C_1 \cdot [\textit{wetted area(time=t)}]^{C_2} \qquad (8.9)$$

where:

$$wetted\ area(time=t) = wetted\ area(at\ time=0) \times \left\{1 - C_3\left[1 - \frac{LiqVol(time=t)}{LiqVol(time=0)}\right]\right\} \qquad (8.10)$$

Adiabatic Mode

Figure 8.20



The **Adiabatic** depressuring mode brings up the **Adiabatic** group box which contains the following fields:

Isentropic Efficiency

The depressuring utility does not take momentum effects into account when doing the energy balance. This can have a large effect on the final temperatures (15 degrees plus, when you dealing with very fast flows). If you are depressuring in **Adiabatic** mode, you need the **isentropic efficiency %**, which, if you choose the correct value, can bring the temperatures into line. The following table gives a description of the boundary values for this field.

Isentropic Efficiency%	Description
0%	The fluid will not lose enthalpy during the expansion (isenthalpic) but the entropy will change, as this is an irreversible process.
100%	The fluid utilises a required amount of enthalpy during the expansion to maintain constant molar entropy.

The correct value to use is the subject of much debate. Studies have shown that 100% gives a good accord with experimental data for gas filled systems. For liquid containing systems values of 40-70% are generally used. Higher values yield lower final temperatures for the fluid.

Vessel Heat Transfer Area

The **Vessel Heat Transfer Area** is the surface area of vessel or piping in contact with the fluid that will be used to calculate the temperatures by heat balance between the fluid and the wall. For a gas filled vessel this is the total vessel internal surface area. For liquid containing vessels the area exposed to the liquid volume should be used since the Heat Transfer Coefficient for a liquid is much higher than that for a gas. If the user wishes to assume vapour heat transfer properties, they can be estimated by HYSYS (the “wetted area” should be equivalent to the entire vessel area). If the user wishes to liquid heat transfer properties, the user should enter the HTC, and only the wetted surface area should be entered.

Vessel Material Specific Heat Capacity/Vessel Mass

The **Vessel Material Specific Heat** of the metal and **Vessel Mass** are required if you wish to calculate the average vessel wall temperature on depressurisation.

Typical values of vessel specific heat are:

Material	Specific Heat
Mild steel	420 J/kgC
Titanium	523 J/kgC
Stainless steel	510 J/kgC
Copper	385 J/kgC
Aluminium	880 J/kgC
Brass	370 J/kgC

Heat Transfer Coefficient

HYSYS does not take into consideration the heat transfer from the air to the vessel. The only energy transfer is from the mass of the vessel to the fluid.

The **Heat Transfer Coefficient** refers to the metal to fluid interface. No heat transfer between the environment and the vessel is included. This is considered reasonable because the system would see very low temperatures during the depressuring which would mean the vessel exterior would probably be insulated.

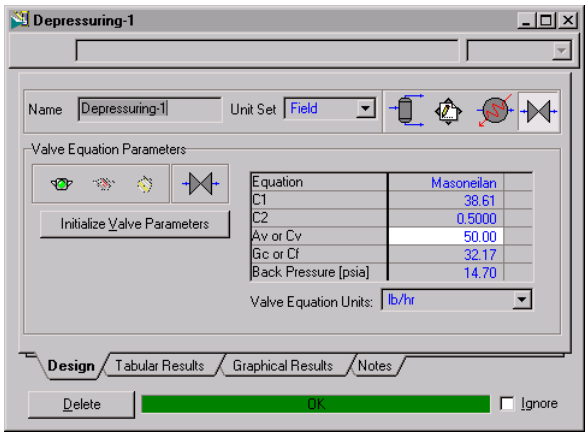
Isothermal Mode

Heat Flux information is not required for this mode.

Valve Parameters Setup Page

Figure 8.21

*Press the **Equation Help** button on for details on the available equations.*



Choosing the Valve Equation

The units specified for the valve equations pertain only to the equations as viewed on this tab; they are not the same as the units used in the simulation.

The Valve Equation is selected from the **Equation** drop down box**,** you have four options:

Equation	Description
Supersonic	$$F = C_1(P_1\rho_1)^{C_2} \quad (8.11)$$ *where:* C_1 = *valve constant*; C_2 = *default power term*; P_1 = *upstream pressure*; r_1 = *upstream density*. Use this for modelling systems when no detailed information is available on the valve. In general the constant C_2 will never vary from 0.5 and the flow through the valve will then be proportional to C_1. One specific exception to this is the modelling of constant flow depressurisation. In this case set C_2 to 0 and C_1 to the required constant flow rate.
Subsonic	$$F = C_1\left[\frac{(P_1 + P_{back})(P_1 - P_{back})}{P_1}\rho_1\right]^{C_2} \quad (8.12)$$ *where:* C_1 = *valve constant*; C_2 = *default power term*; P_1 = *upstream pressure*; P_{back} = *back pressure or valve outlet pressure*; r_1 = *upstream density*. If the pressure in the vessel is such that there is sub critical flow (generally upstream pressure less than twice backpressure), then you have no option but to use the Subsonic Equation. This equation is used in the same instances as the Supersonic equation except when you have subsonic flow. In applying this equation you are required to supply P_{back} or the valve back pressure. By specifying P_{back} to be slightly less than the Relief Pressure, it is possible to have your depressuring analysis cycle between pressure build up and relief. You should ensure a reasonable pressure differential, and you may wish to increase the number of pressure steps for the analysis.

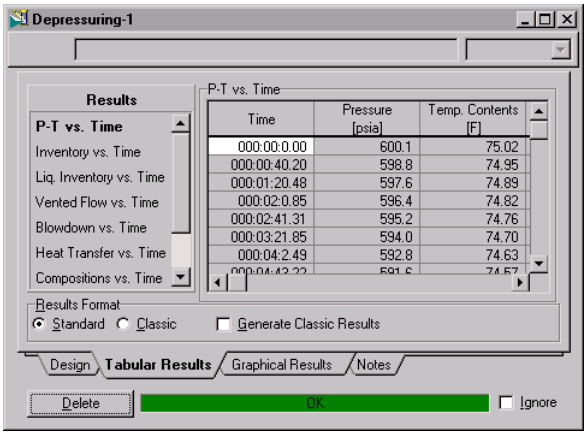
The default values of C_1 are only accurate for the standard SI and Field pressure, mass flow, and density units. If changes are made to these standard units C_1 should be adjusted to ensure consistency. For example, if your units of pressure had been changed to bar, 16.67 should be used for C_1.

Equation	Description
Masoneilan	$$F = C_1 C_v C_f Y_f (P_1 \rho_1)^{C_2} \quad (8.13)$$ *where:* C_1 = 1.6663 (SI default) = 38.86 (Field default); C_v = *valve co-efficient*; C_f = *critical flow factor*; P_{up} = *upstream pressure*; r_{up} = *upstream density*; y = *expansion factor*; $y_f = y - 0.148\, y^3$ (*the max value of* y_f *is* 1) Taken from the Masoneilan catalogue, this equation can be used for general depressuring valves to flare. Often the C_v for a valve is known from vendor data so when Masoneilan is selected the appropriate valve C_1 and C_2 will be automatically set as well as the units. DO NOT change these unless you have a justified reason to do so. HYSYS will estimate a valve C_v based on the input data for the sources and depressuring criteria.The user must adjust this initial C_v in order to ensure that all criteria are fully satisfied.
General	$$F = C_1 43200 A_v K_{term} (g_c P_1 \rho_1 k)^{C_2} \quad (8.14)$$ *where:* C_1 = *dimensionless constant* = 2.625**C*; *C = co-efficient of discharge*; *43200 = dimensional constant*; *Av = valve orifice area*; g_c = *dimensionless constant* = 1.0 *kg.m/N.s*^{2} (32.17 *lb.ft/lb*$_f$*s*^{2}); *k = ratio of specific heats (Cp/Cv)*; P_1 = *upstream pressure*; r_1 = *upstream density*; C_2 = *equation exponent* (= 0.5) This equation is take from *Perry's Chemical Engineering Handbook*. Use it if you know the valve throat area. Note that this equation makes certain limiting assumptions concerning the characteristics of the orifice.

8.6.4 Tabular Results Tab

You may notice a temperature rise in the vessel towards the end of the depressuring. This is due to the fact that the heat transfer from the vessel mass is such that it overcomes the temperature loss due to the expansion of the fluid in the vessel.

Figure 8.22



The **Tabular Results** tab allows you two view the results in two types of printed format.

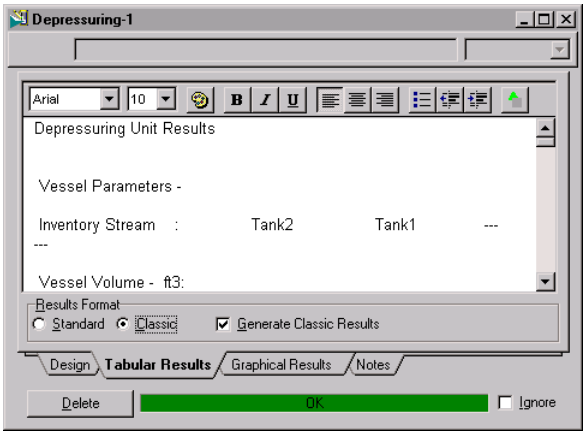
- Standard
- Classic

Standard format is displayed in Figure 8.22. This format is specified by clicking the **Standard** radio button in the Results Format group box. There are eight different sets of results that can be displayed in the table:

- P-T vs. Time,
- Inventory vs. Time,
- Liq. Inventory vs. Time,
- Vented Flow vs. Time,
- Blowdown vs. Time,
- Heat Transfer vs. Time,
- Compositions vs. Time and
- Phys. Props vs. Time.

The Classic format is specified by clicking the **Classic** radio button in the Results Format group box. The Classic format is displayed in Figure 8.23. In order to generate the results for this format the Generate Classic Results check-box must be activate.

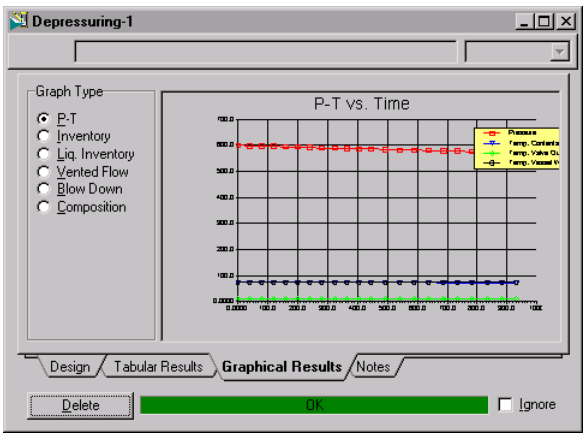
Figure 8.23



The Classic format displays all of the depressuring design information as well as all of the information that can be displayed in the Standard format in one form.

8.6.5 Graphical Results Tab

Figure 8.24



To edit a plot, object inspect the plot area and select ***Graph Control*** *from the menu. For details on manipulating plots, refer to* ***Section 6.4 - Graph Control****.*

The **Graphical Results** tab allows you to examine the results of the depressuring analysis in graphical form. You may observe:

- Pressure - Temperature over time
- Inventory over time
- Liq. Inventory over time
- Vented Flow over time
- Blow Down over time
- Composition over time

8.6.6 Notes Tab

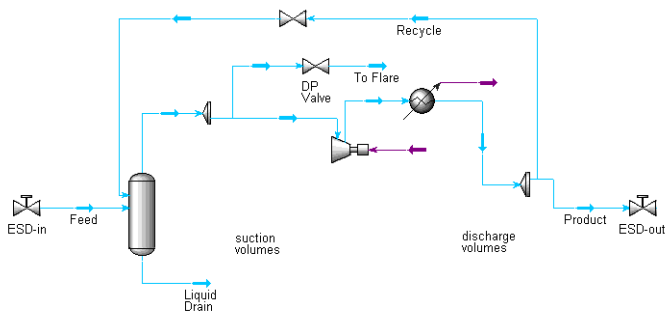
The **Notes** tab provides a text editor where you can record any comments or information regarding the Decompression utility or pertaining to your simulation in general.

8.6.7 Modelling Specific Systems

Modelling A Compressor Loop System

Suppose you wished to model the depressuring of system such as the one shown in Figure 8.25.

Figure 8.25



The loop in question contains 4 input volumes:

Input Volume	Stream	Vessel Volume	Liquid Volume
suction piping	Feed	2 m^3	
suction drum	Feed	5 m^3	0.75 m^3
discharge piping	Product	2 m^3	
exit cooler	Product	1 m^3	

To model the depressuring of this system first add the following streams:

MATERIAL STREAM [Feed]		
Tab [Page]	**Input Area**	**Entry**
Worksheet [Conditions]	Temperature	15 °C
	Pressure	40 bar
Worksheet [Composition]	Methane Mole Frac	0.60000
	Ethane Mole Frac	0.20000
	Propane Mole Frac	0.10000
	i-Butane Mole Frac	0.05000
	n-Butane Mole Frac	0.05000

MATERIAL STREAM [Product]		
Tab [Page]	**Input Area**	**Entry**
Worksheet [Conditions]	Temperature	40 °C
	Pressure	100 bar
Worksheet [Composition]	Methane Mole Frac	0.663472
	Ethane Mole Frac	0.203300
	Propane Mole Frac	0.079220
	i-Butane Mole Frac	0.029061
	n-Butane Mole Frac	0.024947

*The composition of the **Product** stream is simply the vapour phase of the **Feed** stream.*

From the **Tools** menu select the **Utilities** option. This should bring up the **Available Utilities** view. From the list box on the right side of the view select **Depressuring** and press the ***Add Utility*** button.

Fill the **Depressuring** property view as shown below:

UTILITY [Depressuring]			
Tab [Page]	**Input Area**	**Entry**	
Design [Vessel Parameters]	Stream Name	Feed	Product
	Vessel Volume	7 m^3	3 m^3
	Liquid Volume	0.75 m^3	<empty>
Design [Operation Mode]	Mode	Adiabatic	
	Relief Pressure	55.85 bar	
	Depress Time	15 mins	
	Final Pressure	27.93 bar	
	Pressure Steps	30	
	Wetted Area	<empty>	

Note that HYSYS assumes an instantaneous settleout across the system. In practise the blowdown valve will normally open as soon as the ESD valves are closed and consequently the initial flow of gas through this valve will reflect the fluid in the suction drum.

Note that HYSYS assumes an instantaneous settleout across the system. In practise the blowdown valve will normally open as soon as the ESD valves are closed and consequently the initial flow of gas through this valve will reflect the fluid in the suction drum.

UTILITY [Depressuring]		
Tab [Page]	**Input Area**	**Entry**
Design [Heat Flux Parameters]	Isentropic Efficiency	70%
	Vess. Specific Heat	420 kJ/kg/C
	Vessel Mass	6000 kg
Design [Valve Parameters]	Valve Equation	Masoneilan
	C1	If Upstream Pressure units are in *kpa* 1.6663, if the Upstream Pressure units are in *bar* 16.663

Leave the **Cv** on the **Valve Eqns** tab as initially estimated by HYSYS and click ***Calculate***.

Sizing the Relief Valve

HYSYS will run until either the final pressure, pressure calculation steps or time limit is reached. When this is done, examine the **Tabular Results** tab and verify that your final pressure has been reached. If the time limit is reached but not the final pressure, then increase the Cv of the valve. If the final pressure is reached before the time limit, then decease the valve Cv. Using a quick trial and error process you should arrive at a Cv of **2.07** as being the required valve size.

Note that for gas filled vessels the depressuring time will be approximately *inversely proportional* to the Cv. However, for two phase systems, the liquid behaviour during depressurisation may complicate the blowdown volume significantly and this relationship may not be valid.

The depressuring utility can be often slow due to the number of flash calculations performed. If the fluid you are depressuring contains a number of components that have zero flow or very small flow these will add to the calculation time.

For example if you are depressuring the compressor circuit shown above, the stream definition will contain all the components used in the simulation including any hypotheticals, NBP cuts and water. In practice the actual composition at this point in the process will consist of the lighter hydrocarbons only. If you wish to do repetitive depressuring evaluation consider removing all the components with insignificant content.

Note that selecting higher isentropic efficiencies in the adiabatic mode will slow down the calculations.

8.7 Derivative

The **Derivative** utility is a component of the HYSYS.RTO real-time optimization package available as a plug-in to the basic HYSYS software package. The **Derivative** utility is one of two utilities used by HYSYS.RTO to provide the primary interface between the flowsheet model and the solver. Their primary purpose is to collect appropriate optimization objects which are then exposed to solvers to meet a defined solution criteria.

Please refer to the HYSYS.RTO User's manual for details concerning the use of this utility. This manual details all features and components related to the HYSYS real time optimization package.

If your current HYSYS version does not support RTO, contact your local Hyprotech representative for more details.

8.8 Envelope

The **Envelope** utility allows you to examine relationships between selected parameters, for any stream of known composition, including streams with only one component. Vapour-Liquid Envelopes may be plotted for the following variables:

- Pressure-Temperature
- Pressure-Volume
- Pressure-Enthalpy
- Pressure-Entropy
- Temperature-Volume
- Temperature-Enthalpy
- Temperature-Entropy

For the Pressure-Temperature envelope, quality lines and a hydrate curve can also be added to the plot. The remaining curves allow the inclusion of Isocurves (Isotherms or Isobars).

Since the **Envelope** is calculated on a dry basis, you must be careful when applying the utility to multi-component mixtures that contain H_2O or any other component which can form a second liquid phase. Note also that the Envelope is restricted to the **Peng Robinson** and **Soave Redlich Kwong** equations of state.

The **Envelope** utility will be illustrated using stream **FEED** (Peng Robinson property method):

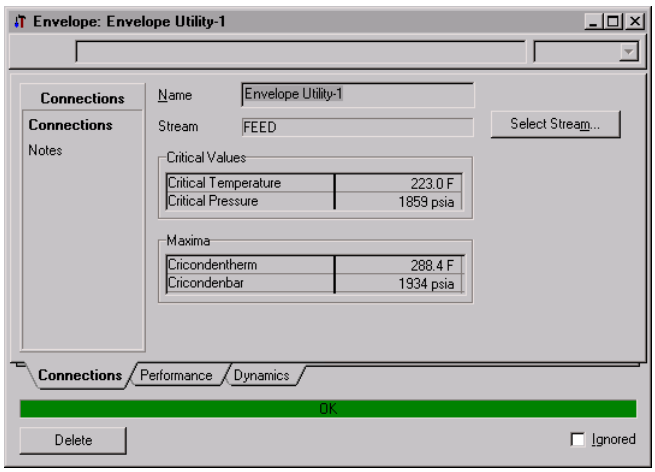
MATERIAL STREAM [FEED]		
Tab [Page]	**Input Area**	**Entry**
Worksheet [Conditions]	Temperature	60.0000 °F
	Pressure	600.0000 psi
	Molar Flow	100.0000 lbmole/hr
Worksheet [Composition]	Nitrogen Mole Frac	0.0069
	CO2 Mole Frac	0.0138
	Methane Mole Frac	0.4827
	Ethane Mole Frac	0.1379
	Propane Mole Frac	0.0690
	i-Butane Mole Frac	0.0621
	n-Butane Mole Frac	0.0552
	i-Pentane Mole Frac	0.0483
	n-Pentane Mole Frac	0.0414
	n-Hexane Mole Frac	0.0345
	n-Heptane Mole Frac	0.0276
	n-Octane Mole Frac	0.0206

To add the Envelope utility, select **Utilities** under **Tools** in the Menu Bar, highlight **Envelope**, and select the ***Add Utility*** button.

8.8.1 Connections

You may attach a stream from the Object Navigator by pressing the ***Select Stream*** button. For this example, select stream **FEED**. The **Plots** tab will immediately be shown for the selected stream.

Figure 8.26



On the **Connections** page, the Critical Temperature and Pressure are calculated, as well as the Cricondentherm and Cricondenbar.

The **Dynamics** tab allows you to control how often the utility gets calculated when running in Dynamic mode (refer to the Dynamics sub-heading in **Section 8.1 - Boiling Point Curves** for more information).

Note that you may ignore this Utility during calculations by selecting the **Ignored** check box. HYSYS will disregard the utility entirely until you restore it to an active state by clearing the check box.

8.8.2 Plots

On the **Plots** tab, select a radio button in the **Envelope Type** group to display the associated envelope.

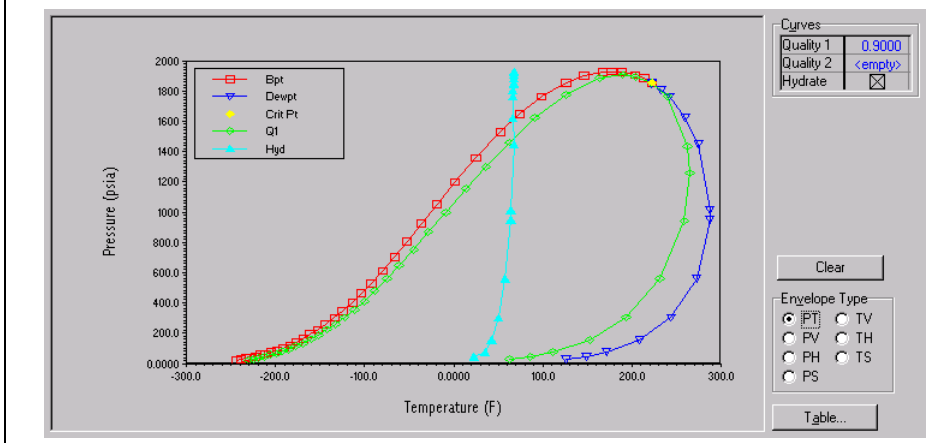
By pressing the ***Table*** button, you can access the envelope results in tabular format. Simply select an option from the drop down list. All Isocurves and Quality lines associated with the individual envelopes will be transferred to the table.

The various available envelopes will now be covered in more detail.

Pressure-Temperature Envelope

Select the **PT** envelope in the **Envelope Type** group.

Figure 8.27



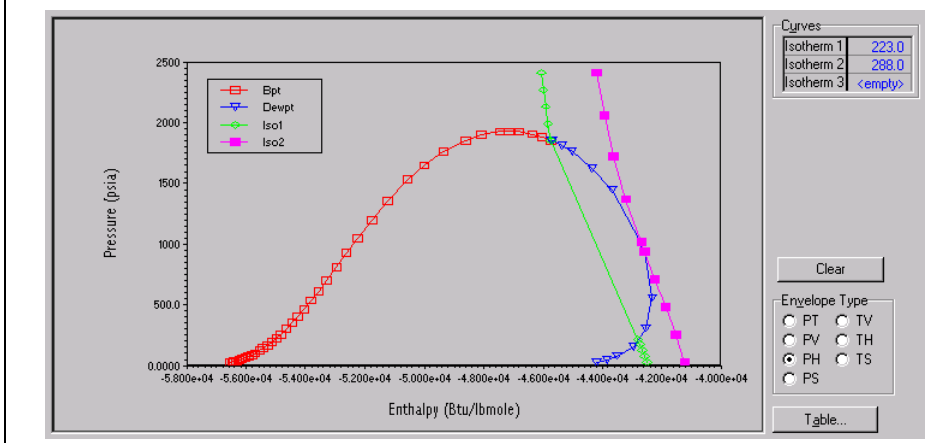
The Vapour-Liquid envelope for a quality of 1.0 is automatically displayed. This is actually represented by two curves; one with a vapour fraction of 1.0 and the other having a liquid fraction of 1.0. These curves meet at the stream critical point. You may plot additional envelopes for different qualities simply by typing the desired quality (between 0 and 1) in the **Quality 1** and **Quality 2** input cells. In this example, the envelope for a quality of 0.9 is shown. A quality of 0.9 is represented by two curves; one with a vapour fraction of 0.9 and the other having a liquid fraction of 0.9.

Activate the Hydrate check box to have HYSYS calculate and display the hydrate temperature curve for pressures up to the cricondenbar. Further discussion on the calculation of hydrates can be found in Section 8.9 - Hydrate Formation.

You may clear all non-default curves by selecting the ***Clear*** button.

8.8.3 PV-PH-PS Envelopes

Figure 8.28



If you select the **PV** radio button, the Pressure-Volume Envelope is displayed. Selecting the **PH** radio button will display the Pressure-Enthalpy Envelope, while selecting the **PS** radio button will display the Pressure-Entropy Envelope.

For each of these Envelopes, you may display a maximum of three Isotherms (constant temperature curves) by entering values in the **Curves** group.

Shown in Figure 8.28 is the Pressure-Enthalpy envelope for stream **FEED**, with 223°F and 288°F Isotherms.

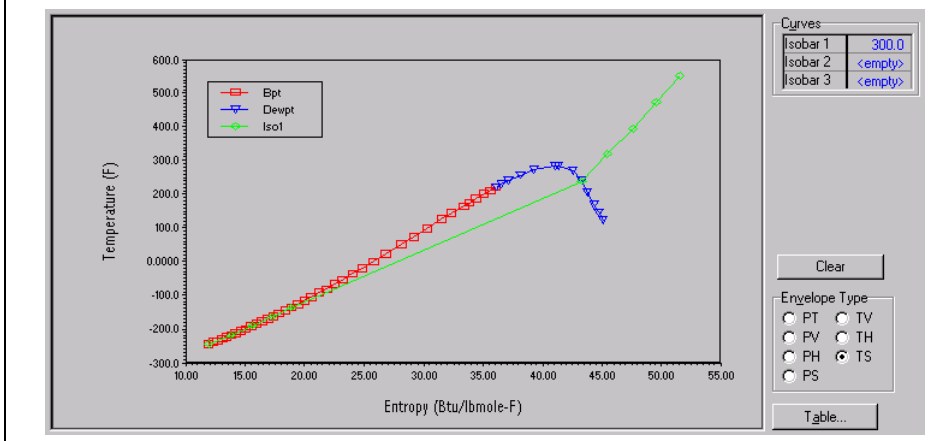
You may clear all curves (except the default) at any time by selecting the ***Clear*** button.

8.8.4 TV-TH-TS Envelopes

If you select the **TV** radio button, the Temperature-Volume Envelope is displayed. Selecting the **TH** radio button will display the Temperature-Enthalpy Envelope, while selecting the **TS** radio button will display the Temperature-Entropy Envelope.

For each of these Envelopes, you may display up to three Isobars (constant pressure curves). Simply enter the desired pressure(s) in the **Curves** group.

Figure 8.29



Shown in Figure 8.29 is the Temperature-Entropy envelope for stream **FEED**, with a 300 psi Isobar.

As before, you may clear all non-default curves at any time by selecting the ***Clear*** button.

8.9 Hydrate Formation

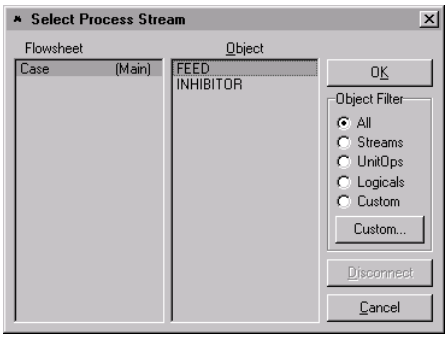
Hydrates formation prediction is restricted to the Peng Robinson and Soave Redlich Kwong equations of state.

The **Hydrate Formation** utility will calculate the incipient solid formation point for gas hydrates. The predictive models are based on fundamental thermodynamic principles and use equation-of-state generated properties in calculating the equilibrium conditions. These predictive models can therefore be applied to various compositions and extreme operating conditions with a greater degree of reliability than one might expect with empirical expressions or charts. Note also that a hydrate curve can be plotted with the Envelope utility.

8.9.1 Stream Settings

You can set the name of the utility as well as the stream for which this utility will be calculated. You may select a stream from the Object Navigator, which is accessed by pressing the ***Select Stream*** button.

Figure 8.30



The **Dynamics** tab allows you to control how often the utility gets calculated when running in Dynamic mode (refer to the Dynamics sub-heading in Section 8.1 - Boiling Point Curves for more information).

Note that you may ignore this Utility during calculations by selecting the **Ignored** check box. HYSYS will disregard the utility entirely until you restore it to an active state by clearing the check box.

For this example we are defining two streams, **FEED** and **INHIBITOR**, as shown below (use the **Peng Robinson** property method). The Hydrates Formation utility will be used to determine the hydrate forming conditions before and after injection of the methanol inhibitor

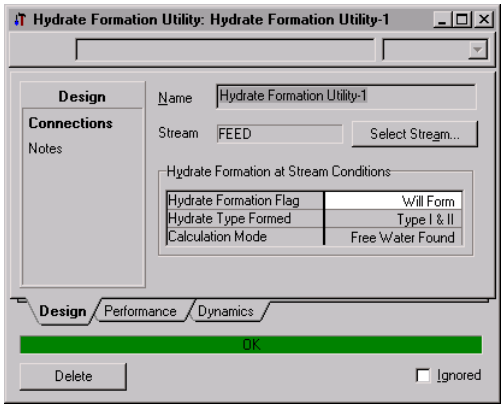
.

MATERIAL STREAM [FEED]		
Tab [Page]	**Input Area**	**Entry**
Worksheet [Conditions]	Temperature	5.0000 °C
	Pressure	7000.0000 kPa
	Molar Flow	50.0000 kgmole/hr
Worksheet [Composition]	Nitrogen Mole Frac	0.0590
	Methane Mole Frac	0.7091
	Ethane Mole Frac	0.0468
	Propane Mole Frac	0.0192
	n-Butane Mole Frac	0.0078
	n-Pentane Mole Frac	0.0078
	CO2 Mole Frac	0.1404
	H2O Mole Frac	0.0099
	Methanol Mole Frac	0.0000

MATERIAL STREAM [INHIBITOR]		
Tab [Page]	**Input Area**	**Entry**
Worksheet [Conditions]	Temperature	5.0000 °C
	Pressure	7000.0000 kPa
	Molar Flow	10.0000 kgmole/hr
Worksheet [Composition]	Nitrogen Mole Frac	0.0000
	Methane Mole Frac	0.0000
	Ethane Mole Frac	0.0000
	Propane Mole Frac	0.0000
	n-Butane Mole Frac	0.0000
	n-Pentane Mole Frac	0.0000
	CO2 Mole Frac	0.0000
	H2O Mole Frac	0.2000
	Methanol Mole Frac	0.8000

8.9.2 Hydrate Formation at Stream Conditions Group

Figure 8.31



The Hydrate Formation status at the current stream conditions are shown on the **Connection** tab.

Hydrate Formation Status	Description
Hydrate Formation Flag	Displays the status of hydrate formation. There are two possibilities, **Will Form** and **Will NOT Form**.
Hydrate Type Formed	Will display the types of Hydrate formed. In this example, for stream **FEED**, hydrate types I and II are formed. It is possible that Ice will form first, in which case HYSYS will display the message **Ice Forms First** in the appropriate field. If the temperature is higher than the formation temperature, then No Types will appear in this field.
Calculation Mode	Possibilities are Vapour Phase, Free Water Found and Assume Free Water. HYSYS has the capability of predicting the incipient solid formation point for systems consisting of gas hydrates in equilibrium with a free-water phase or for systems without a free-water phase. Note that it is not necessary for a free-water phase to be present for hydrate formation. For either case, the correct model will be used to predict the incipient point for solid hydrates. If water is not specified as a component, HYSYS assumes the stream to be saturated with water.

The only requirement for hydrate formation is that some water must be present in either the vapour or condensed hydrocarbon phase with hydrate forming components. Once favourable pressure and temperature conditions are reached (high pressures or low

temperatures), the mixture of hydrate-forming molecules and water molecules will form a non-stoichiometric solid phase. Note that these conditions can be well above the freezing point of water or well before the point where free water or ice would drop out.

The hydrate formers are limited to molecules that are small enough to fit into the cavities formed by the host water lattice structure. These include low molecular weight paraffinic hydrocarbons up to n-butane, some olefins, and some of the smaller non-hydrocarbon components such as carbon dioxide, nitrogen and hydrogen sulphide.

In many cases, you will run into processing situations where there will be free water present and hydrate formation would be a serious concern, such as after chilling wet reservoir fluids, or in cryogenic processing with very low water content (undersaturated vapour). A completely rigorous approach is used within HYSYS for determining at what point a solid hydrate phase forms. As such, you do not have to be concerned with specifying the proper model for the solid prediction.

Note that if water is not included as a component, the utility will assume the stream to be saturated with water and will use the correct model for predicting the hydrate point.

There are instances where water is specified as a component and the stream is undersaturated such that no free water forms during the search. In this case, a different model specifically developed for predicting the incipient point for hydrate formation in equilibrium with a specified water content will be used. As in the other model, it can be used for determining the hydrate point in the vapour or condensed hydrocarbon phase.

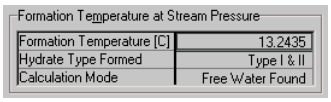
The model used for predicting the incipient hydrate point for hydrates in equilibrium with free water is based on the original equilibrium model proposed by van der Waals and Platteeuw[1] and later modified by Parrish and Prausnitz[2]. The same model has been incorporated and enhanced by Hyprotech for its hydrate predictions. The equation of state is used to predict the properties of the hydrate-forming components in equilibrium with the solid hydrate phase. This can be used for predicting hydrate formation in the vapour or condensed hydrocarbon phase, commonly referred to as vapour and liquid phase hydrates respectively. A detailed description of the mechanisms involved in hydrate formation and the concepts used in the formulation of the predictive model are given in the above references.

A similar approach is used for predicting the hydrate point for systems without free water present, with the exception that the fugacity of water

in the corresponding hydrocarbon phase and the hydrate lattice are taken into account. A conceptual description of the model is given in the articles by Sloan, Khoury and Kobayashi[3] and Ng and Robinson[4].

8.9.3 Formation Temperature at Stream Pressure

Figure 8.32

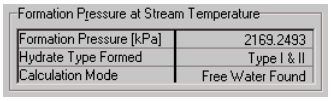


If you select the **Hydrate P/T** page, you will see the Formation Temperature at which hydrates are formed, at the stream pressure. The Hydrate Type and Calculation Mode are also shown for the hydrate which would form at this formation temperature. The Hydrate Types and Calculation Modes are discussed in the previous section.

For stream **FEED**, the Formation Temperature at the Stream Pressure of 7000 kPa is 13.24°C.

8.9.4 Formation Pressure at Stream Temperature

Figure 8.33



On the same page, the Formation Pressure at which hydrates are formed is shown, at the stream temperature. The Hydrate Type and Calculation Mode are also shown for the hydrate which would form at this formation pressure. The Hydrate Types and Calculation Modes are discussed in the Hydrate Formation section.

For stream **FEED**, the Formation Pressure at the Stream Temperature of 5°C is 2169 kPa.

8.9.5 Hydrate Inhibition

To avoid or inhibit the formation of hydrates, you have a number of available options. You can either set the operating conditions to be outside the predicted equilibrium curve for hydrates, or inject inhibitor solvents such as glycols or alcohols to suppress the formation of hydrates. The solvents serve as antifreeze agents and depress the freezing conditions of hydrates.

To inhibit the formation of hydrates of a given stream in the Flowsheet, you must install a stream which contains the solvent (e.g., either methanol or glycol). Use the **MIXER** operation to mix it with the process stream, and then access the Hydrate utility to find the new solid hydrate formation condition. HYSYS will also report if the solid solvent phase forms before the hydrates do (i.e., solid methanol or glycol solution). These equilibrium conditions have all been fitted from known phase diagrams. The eutectic point formed by the solvent mixture results in a solid methanol or glycol phase if a high concentration is used.

In setting up a Flowsheet for hydrate inhibition, you must ensure that the conditions of the solvent injection stream are all sufficiently defined (i.e., Temperature, Pressure, flow rate and composition) so that the property package can flash the mixed stream. As a result of solvent injection, the hydrate-forming conditions will be reduced due to association of the inhibitor with the water in the current phase (i.e., vapour or liquid). Also note that since three phase thermodynamics are used to perform the flash calculation, the phase distribution of the components, including water and the solvent, will be calculated rigorously. Therefore, solvent losses in the hydrocarbon liquid and vapour phases are properly taken into account.

The PR equation of state was not originally designed for non-ideal components such as methanol and glycols. You should ensure that the resulting distribution of the components in all phases is satisfactory, especially if three phases exist. The solubility of methanol in the hydrocarbon and aqueous phases has been optimized with the PR Equation of State for the methanol-HC-water VLE. You may wish to further adjust the PR interaction parameters to meet your own specifications.

Overall, this approach should be more accurate than using Hammerschmidt's equation which was developed more for dilute solutions of antifreeze agents. The Hammerschmidt equation applies only for typical natural gas mixtures and for solute concentrations less

than 20 mole per cent. Although it has been applied for cases beyond this region with reasonable success, this has been attributed to a number of compensating factors. For validation of this model, refer to **GPA Research Report RR-66**.

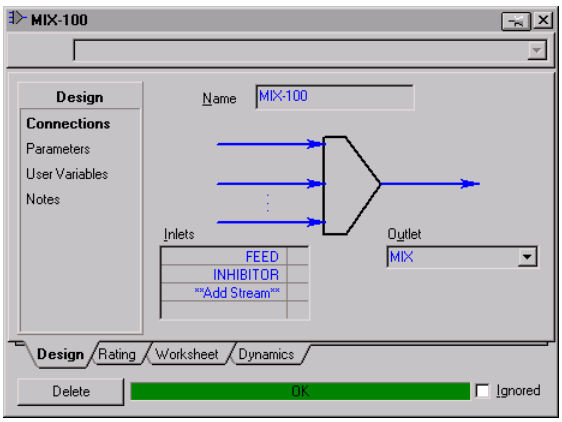
8.9.6 Hydrate Inhibition Example

In this example, a methanol stream of 80 mole percent methanol and 20 mole percent water (stream INHIBITOR) is injected into stream FEED at a molar flow rate of 10 kgmole/h and is then separated in a three-phase separator. The temperature of the streams for this part of the example is -5°C. The solid utility is used to determine the hydrate forming conditions after the injection of the methanol inhibitor.

For the remainder of this example, set the temperature of streams FEED and INHIBITOR to -5 °C.

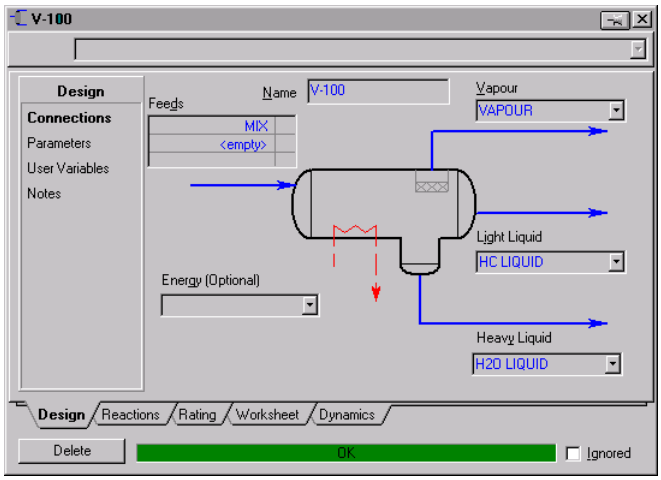
Install the **MIXER** operation to combine streams **INHIBITOR** and **FEED** to produce Stream **MIX**.

Figure 8.34



Now install the **THREE-PHASE SEPARATOR**, completing the **Connections** page as shown. All other defaults will be used.

Figure 8.35



The hydrate temperature of streams **FEED** and **Vapour** can be calculated to determine the effectiveness of the inhibitor.

The hydrate formation temperature of stream **FEED** is 13.24°C. The hydrate formation temperature of stream **Vapour** is -41.6°C. The methanol reduces the hydrate temperature by about 55°C.

8.10 Parametric Utility

The Parametric utility is a set of tools for building a Parametric model (PM) within the HYSYS environment. The utility integrates *Neural Network* (NN) technology into its framework. The major function of the Parametric Utility is to approximate an existing HYSYS model with a Parametric model.

Using a Parametric model with neural network capability to approximate a HYSYS model will significantly improve the robustness of the model and reduce its calculation time thereby improving overall on-line performance. The accuracy of the model will depend upon the data available and the type of model being approximated

The object of analysis can be a collection of unit operations, an entire flowsheet, or a number of selected variables. Using input and output

data sets as training data, the neural network algorithm will determine the values of the Parametric model parameters. This step is called *training* but can also be referred to as *regression* or *identification*.

8.10.1 Neural Networks

Neural Networks provide a cost effective modeling tool, and can extend the capabilities of traditional statistics, modeling and control. They can be applied in both linear and non-linear systems where first principles modeling is costly or difficult.

Neural Networks provide very flexible and powerful techniques for data analysis, and can be used for:

- Dynamic and Static Process Modeling
- Nonlinear and Adaptive Control
- Inferential Predictions
- Time Series Prediction
- Multivariate Pattern Recognition

8.10.2 Variables

*Refer to Section 10.3 - Parametric Unit Operation of the **Steady State Modelling** manual for details on the **Parametric Unit Operation**.*

The parameters of the Parametric model are determined either through HYSYS simulation runs or based on historical plant data (the latter also requires the use of the Parametric unit operation). The following terminology is used to describe the variable types used in the Parametric model:

Observable Variables

Observable variables can be either input or output variables within the HYSYS PFD Model. When HYSYS is used to generate training datasets for the Parametric model, a number of simulation runs will be performed. During the simulation run, the simulation solution engine will calculate each operation in the HYSYS PFD. The observable variables are the HYSYS variables whose values are known and used as training data when calculating the Parametric model.

It is important to note that observable input and output variables may each include both input and output stream variables. A HYSYS model parameter with a varying value can be either an observable input variable or an observable output variable within the Parametric model.

Manipulated Variables

The manipulated variables are the variables being modified in the Parametric Utility and are obtained from the HYSYS PFD model simulation.

Training Variables

Training variables are a combination of both the Observable and Manipulated variables used to develop the Parametric model. The term *training* refers to the task of using the data sets available as a form of "learning" that in effect, fits the model parameters to the specifications.

The Parametric model approximates the HYSYS model in the sense that, given the same values for the training input variables, the values of the output variables from the Parametric model must be close to the values of the output variables from the HYSYS model.

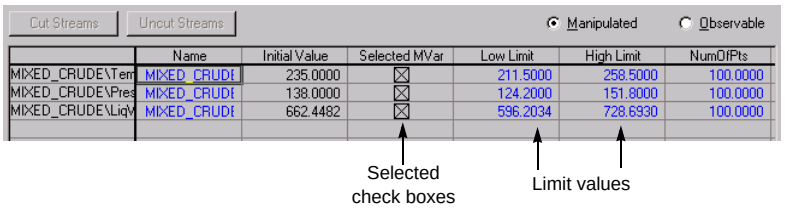
It is important to realize that there are no methods for training neural networks that can "magically" create information that is not contained in the training data. The neural network model is only as good as its training data.

8.10.3 Generalized Procedure

1. The Parametric Utility can be accessed by selecting **Utilities** under **Tools** in the Menu Bar. Choose **Parametric Utility** from the list in the **Available Utilities** view and press the ***Add Utility*** button.
2. On the **Options** tab, change the **Name** of the utility, if desired.
3. Select the **Type of application** for the Parametric model. The choices are:
   - A collection of Unit Operations
   - Whole flowsheet
   - Selection of variables for sensitivity analysis
   - Unknown.
4. Press the **Next** button to continue to the **Configuration** tab.
5. On the **Configuration** tab, press the ***Select Unit Operation*** button to open the **Target Object** view and select the desired flowsheet objects to be used in the Parametric model. The **Object Filter** group filters the available objects by type. The **Objects for Reconciliation** group lists all currently selected objects.
6. Press the ***Accept List*** button once you are satisfied with your selections.

7. The **Configuration** tab view will now list all Manipulated and Observable variables to be used in the model. You can examine a list of both variable types by selecting the **Manipulated** or **Observable** radio button which will display the appropriate table.
8. Use the **Selected** check box within the table and selection buttons to select/un-select or remove variables from the lists (see figure below).
9. Specify **Low Limit** and/or **High Limit** values if you wish to change the range that the Parametric model will work in. The default value is flowsheet value ±10%. (see figure below)

Figure 8.36



10. Press the ***Accept Configuration*** button once you are satisfied with the current variable selection. The Parametric model will now be built based on the selected Manipulated and Observable variables.
11. Press the **Next** button to continue to the **Data** tab.
12. Select the **Create as new** button and select a file name and location to save the training datasets to be created.
13. You may want to modify the **Number of DNS** field before generating HYSYS data. The **DNS** is the number of data sets produced that corresponds to the number of HYSYS runs that will be initiated to generate the data.
14. Press the ***Generate Data*** button which will initiate the HYSYS simulation engine to generate training data for the Parametric model based on the HYSYS model. The trace window will display calculation status information for each simulation run.
15. Access the **Training** tab and select the ***Init/Reset*** button to reset the data window.
16. Press the ***Train*** button which will initiate the Parametric Utility training engine. The data sets generated on the **Data** tab will now be used to "train" the neural network model.
17. The ***View Table*** and ***View Graph*** buttons display comparisons between data generated by the Parametric model and the original data generated by the HYSYS model.
18. Press the ***Next*** button to move to the **Validation** tab. This is the final step in developing a Parametric model.

19. You may modify validation settings by selecting the ***Validation Setup*** button.
20. If you are satisfied with the validation settings, press the ***PM runs*** button to run the Parametric model validation. Since the Parametric model is very efficient, these calculations should be almost instantaneous.
21. Next, press the ***Hysys runs*** button to run the HYSYS model validation. Since these calculations are based on the HYSYS simulation engine, they may take longer to complete.
22. Validation is now complete and you may view the validation results using the ***View Tables*** and ***View Graph*** buttons.
23. On the **Options** tab, selecting the **Embedded into Hysys Flowsheet** checkbox will replace the current HYSYS model within the PFD with the Parametric model.

Modifying variables from the HYSYS flowsheet which are now defined as observable variables according to the Parametric model will not have any effect on the output results (i.e. the flowsheet will not recalculate new values).

When embedding a Parametric model into the HYSYS flowsheet, the HYSYS model will be replaced by the Parametric model. This change will be transparent from the flowsheet side and all replaced streams, unit operations, variables etc. will look the same. However, when manipulated variables are altered on the flowsheet, the calculation engine will run the simulation using the Parametric model rather than the HYSYS model, which will result in exceptionally faster convergence times.

8.10.4 PM Utility View

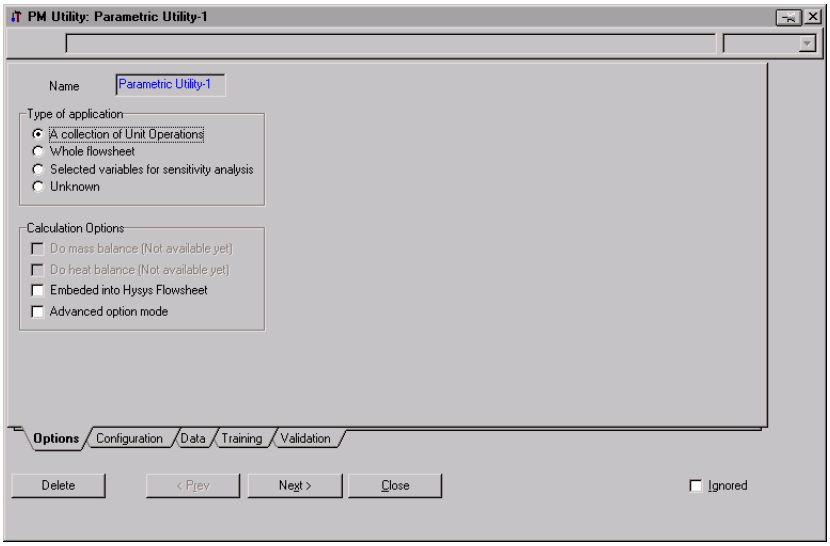
The PM Utility view is composed of several tabs listed in the following table along with a brief description:

Tab	Description
Options	Allows you to specify name, application type and calculation options.
Configuration	Allows you to select HYSYS objects to be included as variables (manipulated or observable) in the Parametric model.
Data	Allows you to generate datasets for use in training the Parametric model.
Training	Initiates the training algorithm to determine parameters for the Parametric model.
Validation	Allows you to validate the Parametric model using validation data.

Options Tab

The **Options** tab allows you to specify the Name, Type of Application and various Calculation Options.

Figure 8.37

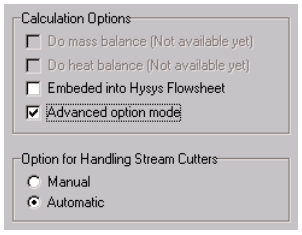


In the **Type of Application** group, various options exist for selecting the components to be approximated with the Parametric model:

- **A collection of Unit Operations** - This option allows the user to select various unit operations from the current HYSYS flowsheet. For the Parametric model to make sense, all unit operations selected must be connected within the flowsheet.
- **Whole flowsheet** - This option selects the entire flowsheet for analysis and thus the entire flowsheet becomes a Parametric model.
- **Selected variables for sensitivity analysis** - This option allows the user to select individual variables from the flowsheet rather than entire unit ops.
- **Unknown** - This option specifies an unknown application type.

Selecting the Advanced option mode allows you to set the handling for stream cutters (Manual or Automatic).

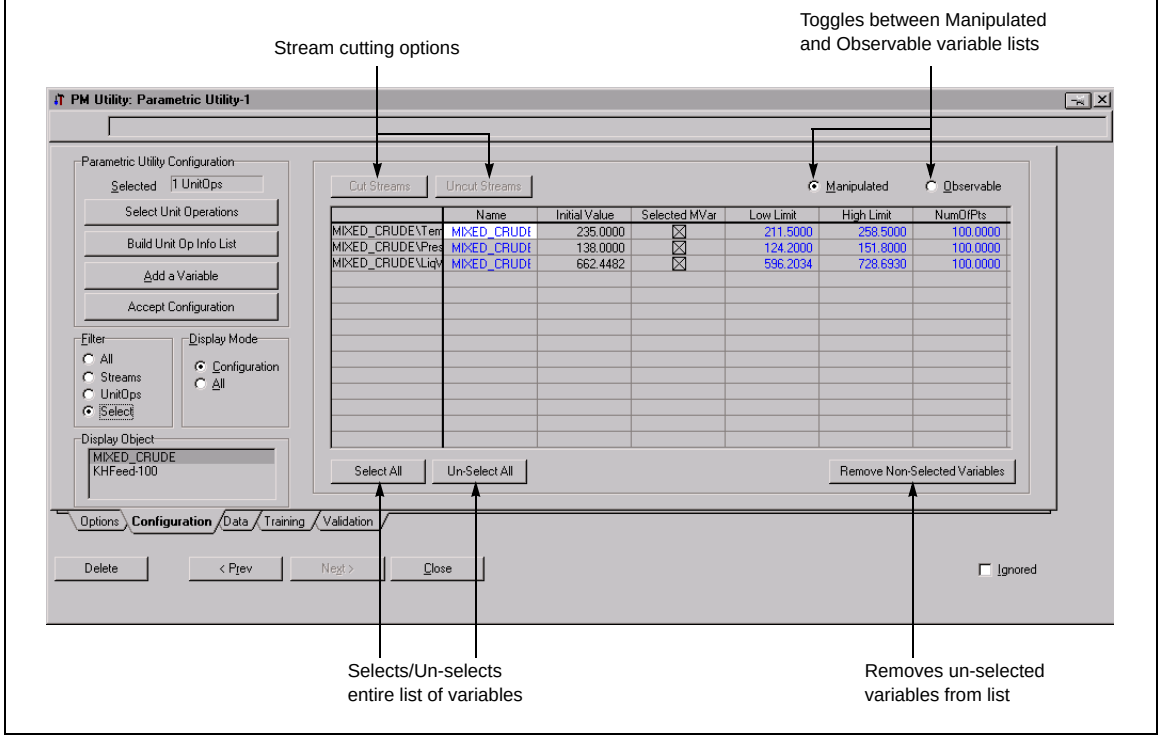
Figure 8.38



Configuration Tab

The **Configuration** tab allows the selection of Manipulated and Observable variables for the Parametric model.

Figure 8.39



Parametric Utility Configuration

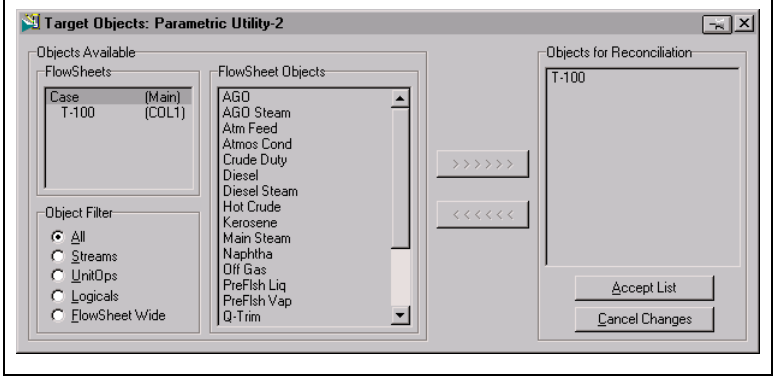
The **Parametric Utility Configuration** group displays the various Parametric model configuration options. The following buttons are available:

Select Unit Operations button

The ***Select Unit Operations*** button opens the **Target Object** view, which allows the selection of objects for reconciliation. These objects can be in the form of streams, unit operations and logicals. The **Filter** group allows objects to be filtered based on category.

The objects selected for use in the Parametric model are listed in the **Objects for Reconciliation** group list.

Figure 8.40



The **Accept List** button accepts the changes and obtains all variables known to the selected objects from the HYSYS flowsheet. Both Manipulated and Observable variables will be listed in the **Configuration** tab view.

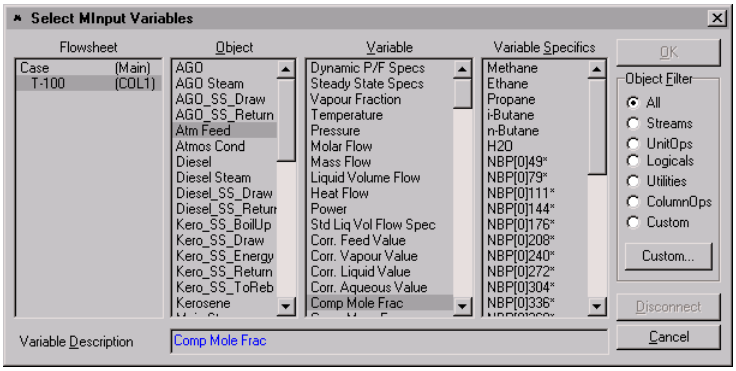
Build Unit Op Info List button

The ***Build Unit Op Info List*** button builds (or updates) a list of variables (Manipulated and Observable) based on the unit operations selected. The operation obtains all variables known to the selected objects in the HYSYS flowsheet and categorizes them as either Manipulated or Observable.

Add a Variable button

The ***Add a Variable*** button allows the user to add variables from the selected unit operations to either the Manipulated or the Observable variable list. If the **Manipulated** radio button has already been selected from the **Configuration** tab, the **Select Input Variable** view will appear and allow the selection of additional Manipulated variables.

Figure 8.41



If the **Observable** radio button were selected, a similar **Select Observable Variables** view will appear and allow the selection of additional Observable variables.

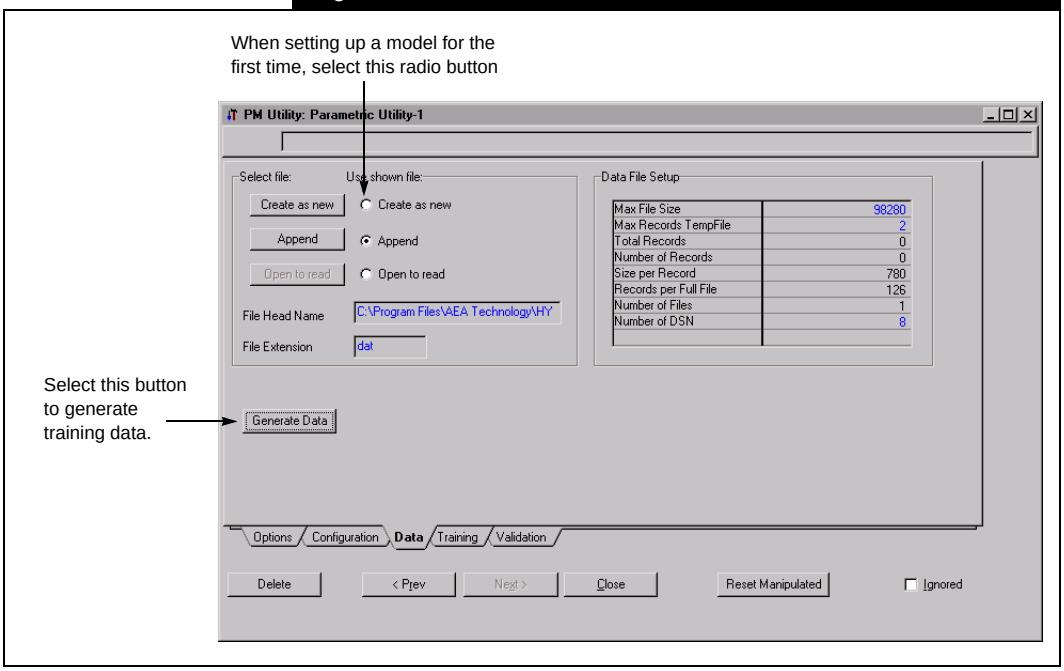
Accept Configuration button

Once you are satisfied with your selections of Observable and Manipulated variables, the **Accept Configuration** button will accept the current variable configuration and allow you to access the **Data** tab.

Data Tab

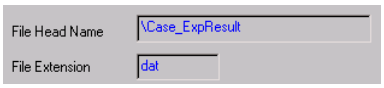
The **Data** tab allows the configuration and generation of input and output data sets for the Parametric model based on HYSYS simulations. Training data sets are generated by using stepwise changes to the manipulated input variables to produce varying output results.

Figure 8.42



When setting up your Parametric model for the first time, click the **Create as new** radio button. Data will be written to an external file based on the default name and location (path) listed:

Figure 8.43



If you wish to choose a specific file name use the ***Create as new*** button that allows you to name and create a new file to store your data.

Later, if you wish to add to the number of datasets used for training, thereby increasing the accuracy of your Parametric model, choose the

Append radio button. Data will be written to a an external file based on the default name and location listed as shown in Figure 8.43.

If your wish to choose a specific file name use the ***Append*** button, which will allow you to name and create a new file to store your data.

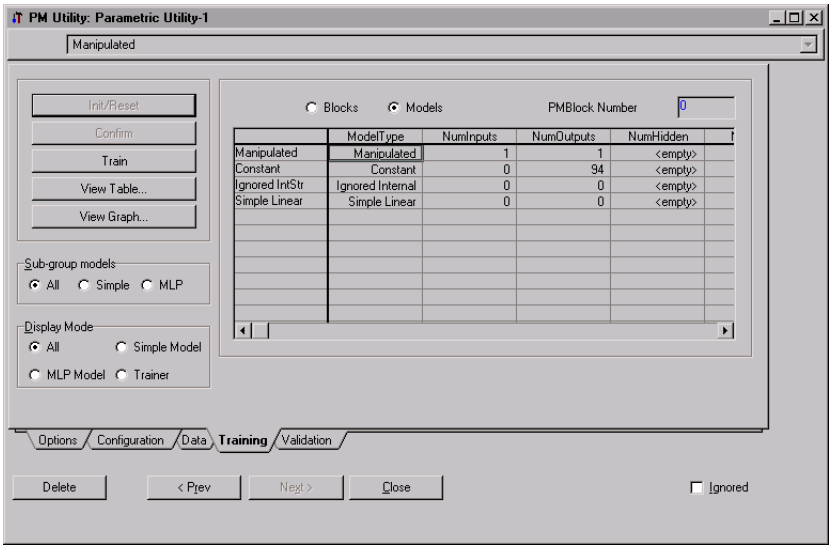
> If you have changed the models configuration, you should not append to existing data sets.

The **Data File Setup** group displays information related to the training data file to be generated by the Parametric Utility. The **Number of DSN** field defines the number of datasets that will be generated using the HYSYS model. Increasing this number increases the likelihood that the Parametric model will be a "good fit" for the flowsheet model, however the data will take longer to generate.

Training Tab

The **Training** tab allows the generation of a Parametric model based on the HYSYS training data. Data sets generated on the **Data** tab are used as training variables. The training algorithm will determine the parameter values of the neural network model based on the input and output datasets. The end result will be a Parametric model which approximates its HYSYS model counterpart.

Figure 8.44

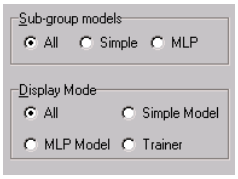


A number of buttons exist for selecting various training options:

Button	Description
Init/Reset	Select this button first before running the training algorithm or whenever you wish to reset the Parametric model.
Confirm	Allows you to confirm the current training configuration.
Train	Initiates the training algorithm to train the neural network based on the data sets generated by the HYSYS model.
View Table...	Allows the viewing of training data in table format. Compares the HYSYS training data with Parametric model data.
View Graph...	Allows the viewing of training data in graphical format. Compares the HYSYS training data with Parametric model data.

The **Sub-group models** group allows filtering of neural network data by either Simple or MLP (Multilayer Preceptrons) or both groups combined (All), The **Display Mode** group displays model data based on the radio button selected.

Figure 8.45

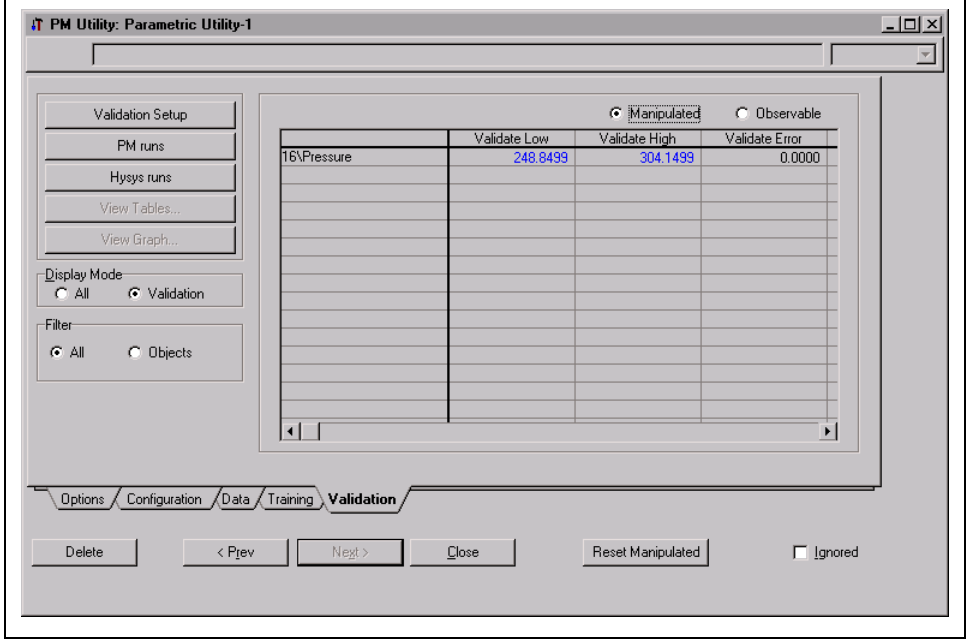


You can also change the number of hidden layers in the MLP and the number of cycles (i.e. number of times the data is presented to the nodes). Changing these may affect the efficiency of your model.

Validation Tab

The **Validation** tab is the final phase in developing a Parametric model. On this tab, validation of the model is performed by generating validation points using both the Parametric model and the HYSYS model and comparing the results.

Figure 8.46

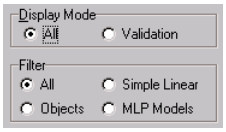


A number of buttons exist for selecting various validation options:

Button	Description
Validation Setup	Allows you to configure validation setup options which include the Number of Validation Points and the Random Speed used in generating the validation points.
PM runs	Runs the Parametric model to generate validation data based on the Parametric model.
Hysys runs	Runs the HYSYS model to generate validation data based on the HYSYS model.
View Table...	Allows the viewing of validation data in table format. Compares the HYSYS validation data with Parametric model data.
View Graph...	Allows the viewing of validation data in graphical format. Compares the HYSYS validation data with Parametric model data.

The **Display Mode** group allows you to display either the entire validation data (**All** radio button) or just the validation range and error (**Validation** radio button). The **Filter** group filters objects based on four radio button selections. Objects are filtered differently depending upon whether they are Manipulated or Observable.

Figure 8.47



8.11 Pinch Utility

Pinch technology is a methodology which is used to optimize the use of process heat exchange and utilities in complicated processes. The HYSYS Pinch utility provides the necessary tools to apply the pinch principles in the design of efficient heat exchanger networks. For further pinch analysis information, refer to the text by Marsland[5].

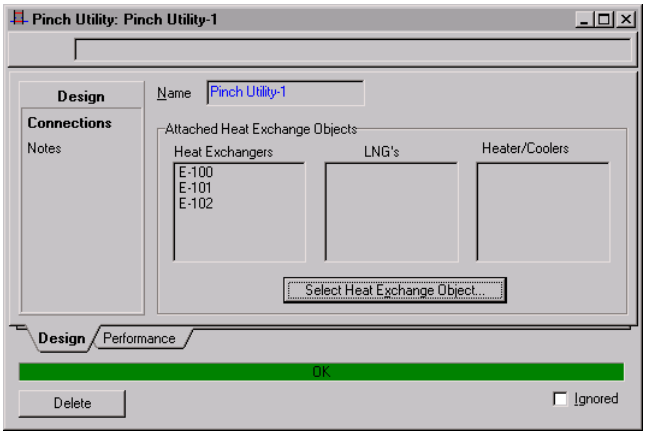
You can attach any combination of heat exchangers, LNG operations, heaters or coolers to the Pinch utility. The only requirement being that each operation is solved so the Pinch calculations can be performed.

8.11.1 Quick Start

As an example of the use of the Pinch Utility, the heat exchanger network example outlined in **Optimizer** chapter of the **Steady State Modelling** guide will be re-visited. If you do not have the case constructed, complete **Section 11.5.1 - Part I: Solving Multiple UA Exchangers** of the **Steady State Modelling** manual.

Access the **Available Utilities** view by selecting **Utilities** under **Tools** in the Menu Bar. Select **Pinch Utility** and press the ***Add Utility*** button. The **Connections** tab of the **Pinch Utility** property view will be displayed.

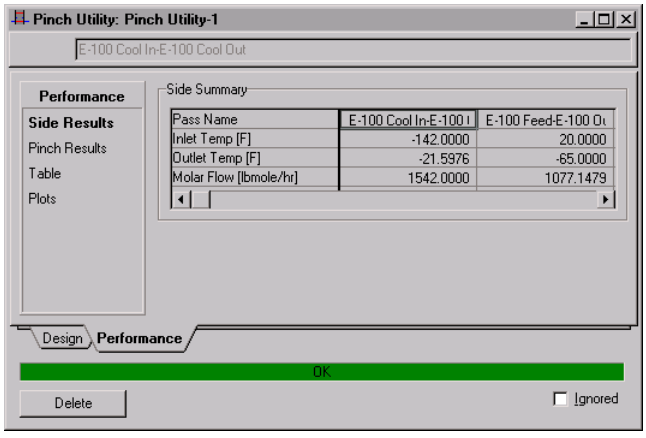
Figure 8.48



On this page, you can attach any combination of heat exchangers, LNG operations, heaters and coolers. Press the ***Select Heat Exchange Object*** button to access the Object Navigator. The object list will be filtered such that only the three heat exchangers in the case are displayed. You will only be able to attach one heat exchanger at a time. When you have attached the three exchangers, the unit names will appear in the **Heat Exchangers** box on the **Connections** tab and the Pinch utility calculations will be completed.

On the **Side Results** tab, examine the Inlet Temperature, Outlet Temperature and Molar Flow of each pass attached to the Pinch utility. In this example, there are six sides since three heat exchangers are used.

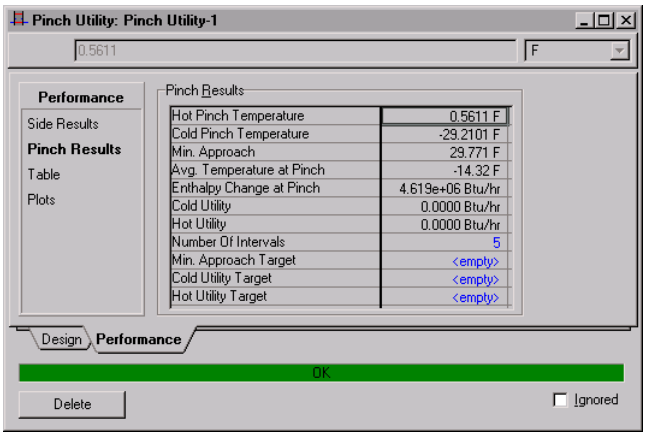
Figure 8.49



You could activate the **Ignored** check box to have HYSYS completely ignore the utility. To restore the utility to an active state, the check box must be cleared.

The various results of the Pinch utility can be examined on the **Pinch Results** tab.

Figure 8.50

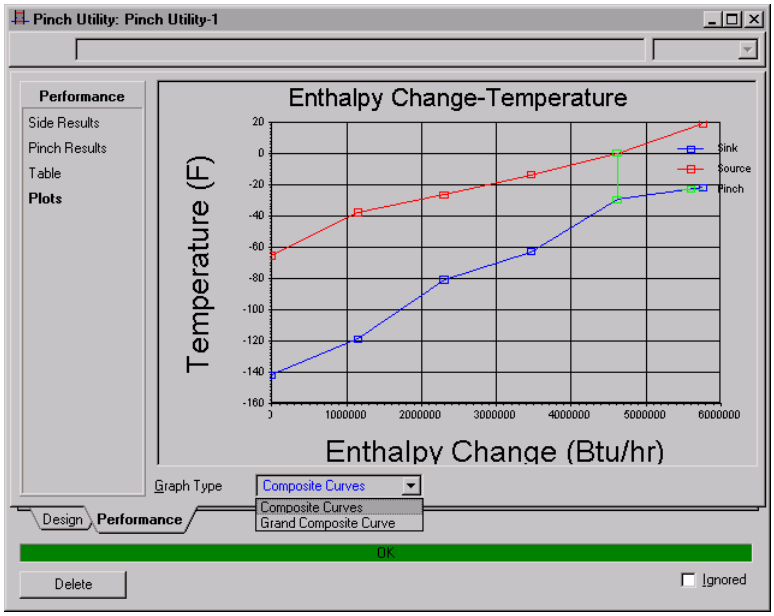


The results which can be examined include:

- Hot Pinch Temperature
- Cold Pinch Temperature
- Minimum Approach - temperature difference between the Hot Pinch and Cold Pinch.
- Average Temperature at Pinch
- Enthalpy Change at Pinch
- Cold Utility
- Hot Utility
- Number of Points - the number of intervals used in the Pinch utility calculations.
- Minimum Approach Target - specifiable minimum approach temperature.
- Cold Utility Target - specifiable cold utility enthalpy value.
- Hot Utility Target - specifiable hot utility enthalpy value.

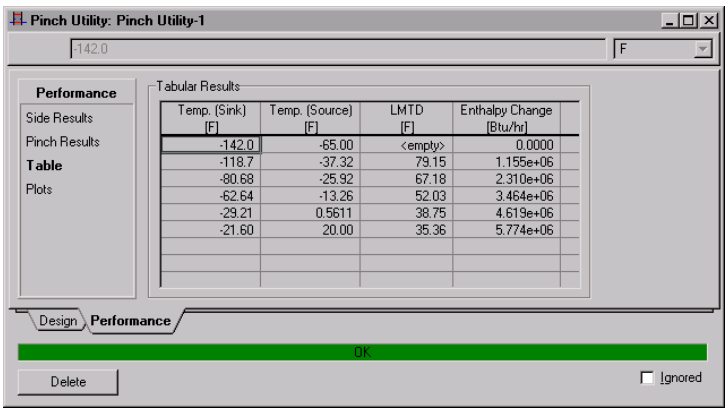
On the **Plots** tab, you can view the Sink and Source **Composite Curves** or the **Grand Composite Curve**. Make your selection from the **Graph Type** drop down list. The Composite Curves for the heat exchanger network is shown below. Notice that the Pinch is also displayed on this plot.

Figure 8.51



The **Table** page shows a tabular report of what is seen on the **Plots** page. You can view temperatures of the Sink and Source, the LMTD and Enthalpy Change for each interval.

Figure 8.52



8.12 Pipe Sizing

With the **Pipe Sizing** utility you can perform design calculations on any of the case streams. Results include pipe schedule, pipe diameter, Reynolds number, friction factor, etc.

8.12.1 Quick Start

For this example, we will be creating a single stream and using the **Pipe Sizing** utility to determine the size the pipe required.

Starting the Simulation

Create a new case and add a fluid package. Select **Peng Robinson** as the property package and include the components listed in the following table:

Property Package	Components
Peng Robinson	Methane, Ethane, Propane, i-Butane, n-Butane, i-Pentane, n-Pentane, Hexane, Heptane, Octane

Access the **Session Preferences** view by selecting **Preferences** on the **Tools** menu. Change the unit set to **Field** units. Create the stream **Feed**.

Name	Feed
Temperature [F]	50.00
Pressure [psia]	480.0
Molar Flow [lbmole/hr]	1000
Comp Mole Frac [Methane]	0.1702
Comp Mole Frac [Ethane]	0.1473
Comp Mole Frac [Propane]	0.1132
Comp Mole Frac [i-Butane]	0.1166
Comp Mole Frac [n-Butane]	0.1066
Comp Mole Frac [i-Pentane]	0.0963
Comp Mole Frac [n-Pentane]	0.0829
Comp Mole Frac [n-Hexane]	0.0694
Comp Mole Frac [n-Heptane]	0.0558
Comp Mole Frac [n-Octane]	0.0417

After installing the **Feed** stream, access the Property Table utility. Choose the **Utilities** option under **Tools** in the Menu Bar. Select **Pipe Sizing** from the **Available Utilities** view and then press the ***Add Utility*** button.

Generalized Procedure

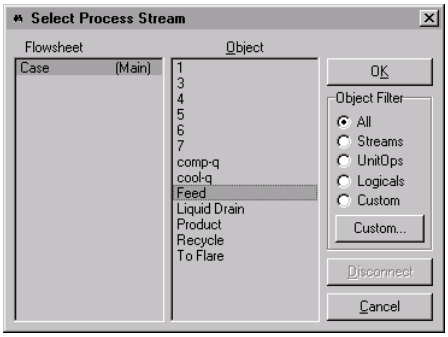
When the **Pipe Sizing** property view appears, you can supply the required input and examine the results of the calculations. A systematic approach to the utility is outlined:

1. On the **Connection** page, change the name of the utility, if desired.
2. Press the ***Select Stream*** button and choose a stream for the analysis from the Object Navigator.
3. Choose the desired **Calculation type**. The options include:
   - **Max. Diameter**
   - **Pressure Drop**
4. Based on the calculation type selected you can now enter the required information to complete the calculation
   - **Max. Diameter -** the input required includes the pipe schedule and the pressure drop in the pipe.
   - **Pressure Drop -** the input required includes the pipe schedule and the pipe diameter.
5. Examine the results of the pipe sizing utility on the **Results** page.

8.12.2 Pipe Sizing View

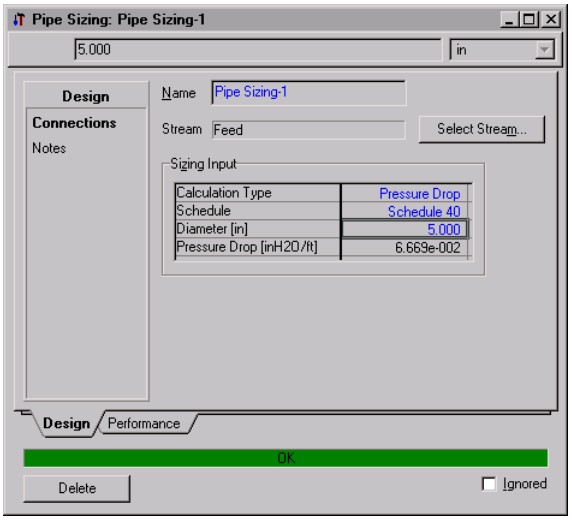
On the **Connection** page you must choose the stream that represent the pipe you wish to size. This is done by clicking the ***Select Stream*** button and selecting one of the available streams from the Object Navigator as shown in Figure 8.53.

Figure 8.53



You may also remove or change the streams to be used by clicking either the ***Remove*** or ***Edit*** buttons.

Figure 8.54

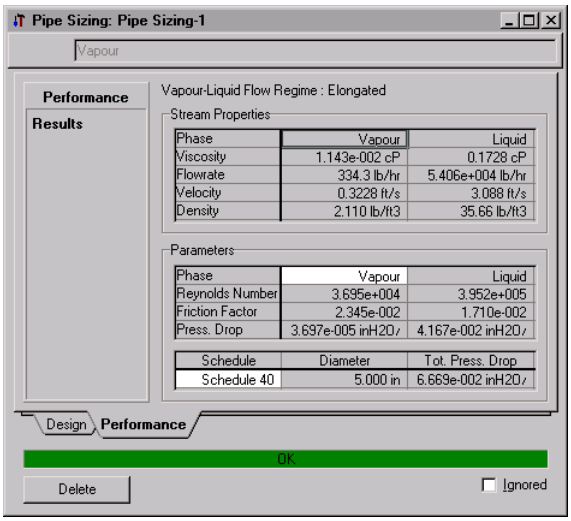


The following fields are available for each stream chosen.

Object	Description
Calculation Type	Allows you to choose between two calculation types: **Max. Diameter** and **Pressure Drop.**
Schedule	Allows you to select a pipe schedule. You are given four selections to choose from: • None • Schedule 40 • Schedule 80 • Schedule 160
Diameter	If you have chosen **Pressure Drop** as your calculation type then you will have to enter a value for the pipe's actual inner diameter. HYSYS will then calculate the pressure drop.
Pressure Drop	If you have chosen **Max. Diameter** as your calculation type then you will have to enter a value for the pressure drop. HYSYS will then calculate the pipe's actual inner diameter diameter.

Once HYSYS has finished calculating the results can be view on the **Results** page of the **Performance** tab.

Figure 8.55



8.13 Property Table

The Property Table utility allows you to examine property trends over a range of conditions in both tabular and graphical formats. Using a stream of known composition, you target two independent variables and their respective ranges of interest. The range of each independent variable is distinct and can be set as either an incremental range or a selection of specific values. Next, you relate which dependent variables are to be displayed at each combination of the independent variables.

8.13.1 Quick Start

For this example, the phase splits for a system of ethanol, water and n-hexane using the **NRTL** activity model is examined. On the **Binary Coeffs** tab of the **Fluid Package** property view, select the **UNIFAC LLE** radio button and press the ***ALL Binaries*** button.

Creating a Fluid Package	
1. Property Package	Activity Model - NRTL
2. Components	Ethanol, H2O, n-Hexane
3. Interaction Parameters	Estimate the unknowns using the UNIFAC LLE estimation technique

MATERIAL STREAM [Stream1]		
Tab [Page]	**Input Area**	**Entry**
Worksheet [Conditions]	Temperature	150.0000 °F
	Pressure	14.7 psi
	Molar Flow	1000.0000 lbmole/hr
Worksheet [Composition]	Ethanol Mole Frac	0.6500
	H2O Mole Frac	0.1500
	n-Hexane Mole Frac	0.2000

After installing **Stream1**, access the Property Table utility. Choose the **Utilities** option under **Tools** in the Menu Bar. Select **Property Table** from the **Available Utilities** view and then press the ***Add Utility*** button.

Generalized Procedure

When the Property Table property view appears, you can supply the required input and examine the results of the calculations. A systematic approach to the utility is outlined:

1. On the **Connections** page of the **Design** tab, change the name of the utility, if desired.
2. Press the ***Select Stream*** button and choose a stream for the analysis from the Object Navigator.
3. Identify one or two independent variables in the **Variable 1** and **Variable 2** (if desired) input cells. The options include:
   - Pressure
   - Temperature
   - Vapour Fraction
   - Enthalpy
   - Entropy

*One of the independent variables must be either **Pressure** or **Temperature**. If the first variable selected is not Temperature or Pressure, the drop down list for the second variable will be limited to **Temperature**, **Pressure** and **Not Set**.*

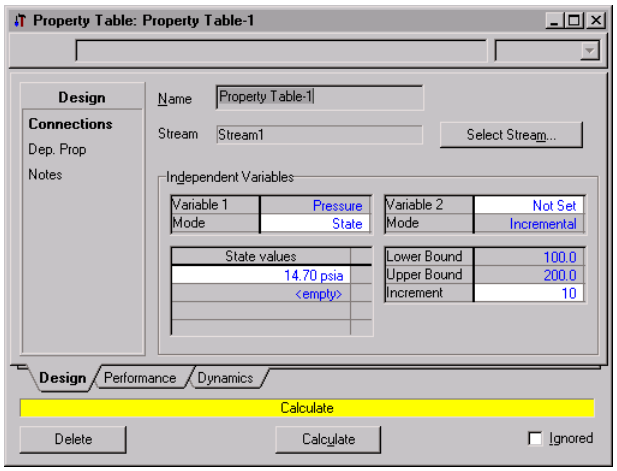
4. You can now select the **Mode** for the independent variable(s). There are two options:
   - **Incremental -** the input required includes the number of increments, and values for the upper and lower bounds. The dependent variable(s) are calculated at each increment within the range.
   - **State -** you can input an unlimited number of specific values for the independent variable.
5. For the incremental variable(s), specify an upper bound, a lower bound and the number of increments.
6. For the state variable(s), specify values for the independent variables in the **State Values** matrix.
7. Switch to the **Dep. Prop** page.
8. Press ***Add*** to select dependent variables from the Object Navigator.
9. Repeat steps 8 until all dependent variables have been identified.
10. Press the ***Calculate*** button.
11. Examine the results of the property table utility in graphical form on the **Plots** page or in tabular form on the **Table** tab.
12. On the **Plots** page, highlight a dependent variable in the **Y Variable** group and press the ***View Plot*** button.

If you select a component related property, such as Component Mole Fraction, you must also select the components that you require for this property.

Independent Variables

For this example, complete the **Connections** page as shown.

Figure 8.56



Stream1 is selected from the Object Navigator as the stream for which the analysis will be completed.

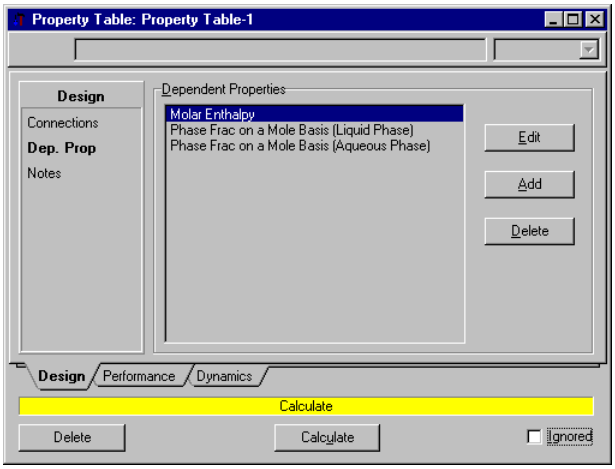
The independent variables are:

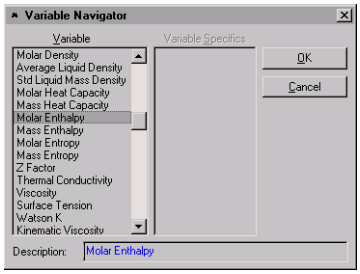
- Pressure, State mode
- Temperature, Incremental mode

The dependent variables will be examined over a temperature range of 100 °F at a constant pressure of 14.7 psi.

Dependent Variables

Figure 8.57





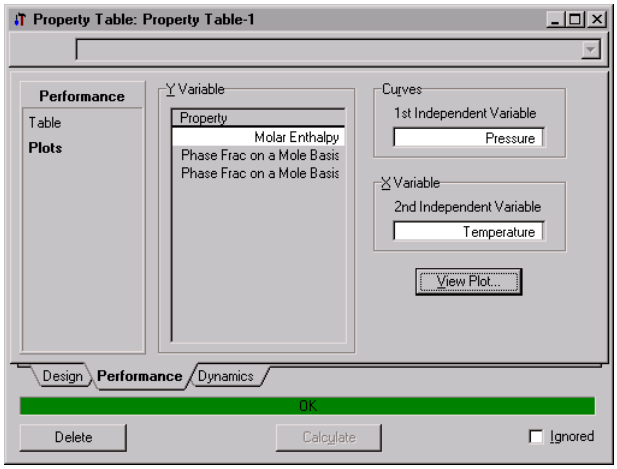
Select the **Dep. Prop** page. On this view, the dependent variables will be selected. The selection of the liquid phase mole fraction is shown. The following dependent properties will be examined in the analysis:

- Molar Enthalpy
- Mole Fraction of Liquid Phase
- Mole Fraction of Aqueous Phase

Start the calculations by pressing the ***Calculate*** button.

Plots

Figure 8.58

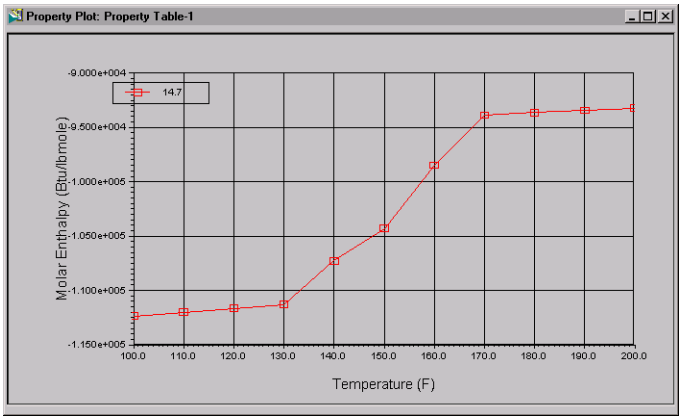


The **Plots** page allows you to display the results of the Property Table utility calculations in a graphical format. Select the **Molar Enthalpy** as the y-variable by highlighting it in the **Y Variable** group. Press the ***View Plot*** button.

*See **Section 6.4 - Graph Control** of the **User's Guide** for details on customizing plots.*

The Molar Enthalpy versus Temperature plot is shown. To make changes to the plot appearance, access the Graph Control property view. Object inspect the plot area and select Graph Control.

Figure 8.59

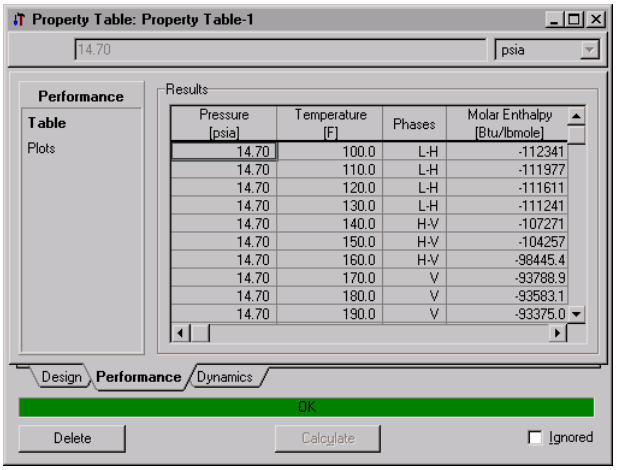


Table

A table listing the results of the property table calculations can be viewed on the **Table** page. The tab, shown in Figure 8.60, lists the independent variables, the dependent variables and the phases present at the given conditions.

The phase column indicates the phases which have been detected at each pair of independent property values. The V indicates vapour, L indicates a light liquid (hydrocarbon rich) phase, and H indicates the presence of a heavy liquid (aqueous) phase.

Figure 8.60



Dynamics

The **Dynamics** tab allows you to control how often the utility gets calculated when running in Dynamic mode (refer to the Dynamics sub-heading in Section 8.1 - Boiling Point Curves for more information).

8.14 Tray Sizing

With the **Tray Sizing** utility you can perform design and rating sizing calculations on part or all of a converged column. Packing or tray information can be specified relating to specific tower internals such as tray dimensions or packing sizes, design flooding and pressure drop specifications. Results include tower diameter, pressure drop, flooding, tray dimensions, etc.

8.14.1 Quick Start

Use of the tray sizing utility requires a converged column. For this example, an 18 stage depropanizer with a full reflux condenser will be installed prior to accessing the utility.

Starting the Simulation

Create a new case and add a fluid package. Select **Peng Robinson** as the property package and include the components listed in the following table:

Property Package	Components
Peng Robinson	Methane, Ethane, Propane, i-Butane, n-Butane, i-Pentane, n-Pentane, Hexane, Heptane, Octane

Access the **Session Preferences** view by selecting **Preferences** on the **Tools** menu. Change the unit set to **Field** units. Create the stream **DeC3 Feed**, which will be the feed for the column:

MATERIAL STREAM [DeC3 Feed]		
Tab [Page]	**Input Area**	**Entry**
Worksheet [Conditions]	Temperature	50.00
	Pressure	480.0
	Molar Flow	1000

MATERIAL STREAM [DeC3 Feed]		
Tab [Page]	**Input Area**	**Entry**
Worksheet [Composition]	Methane Mole Frac	0.1702
	Ethane Mole Frac	0.1473
	Propane Mole Frac	0.1132
	i-Butane Mole Frac	0.1166
	n-Butane Mole Frac	0.1066
	i-Pentane Mole Frac	0.0963
	n-Pentane Mole Frac	0.0829
	n-Hexane Mole Frac	0.0694
	n-Heptane Mole Frac	0.0558
	n-Octane Mole Frac	0.0417

Add a distillation column **DeC3**. The two column specifications include the mole fraction of propane in the bottoms liquid and the mole fraction of isobutane in the overhead vapour.

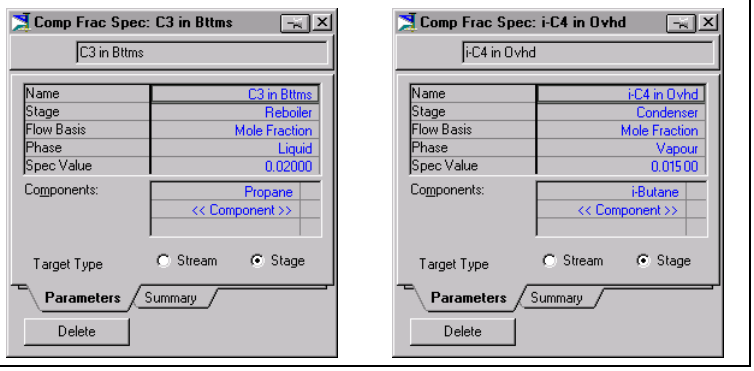
The input required to complete the installation of the column is presented in the following table:

*Please note that in order to directly specify the distillation column DeC3 in the manner described by this table (i.e. tab by tab) the **Use Input Experts** checkbox found on the **Options** page of the **Simulation** tab in the **Session Preferences** property view should be de-activated.*

DISTILLATION COLUMN [DeC3]		
Tab [Page]	**Input Area**	**Entry**
Design [Connections]	No. of Stages	18
	Feed Streams (Stage)	DeC3 Feed (9)
	Condenser Type	Full Reflux
	Ovhd Vapour	Ovhd
	Bottoms Liquid	Bttms
	Reboiler Duty	RebQ
	Condenser Duty	CondQ
Parameters [Profiles]	Condenser Pres.	200 psi
	Cond Delta P	0 psi
	Reboiler Pres.	205 psi

Complete the two column specifications as shown:

Figure 8.61



Once both specifications have been input, switch to the **Monitor** page of the **Design** tab and deactivate the specifications for **Reflux Ratio** and **Overhead Vapour Rate**. Check the **Activate** box for both component fraction specifications that were created. HYSYS will begin solving and the column will converge.

8.14.2 Tray Sizing — Design

The Tray Sizing utility is accessed by selecting **Utilities** under **Tools** in the menu bar. Select **Tray Sizing** from the **Available Utilities** view and press the ***Add Utility*** button.

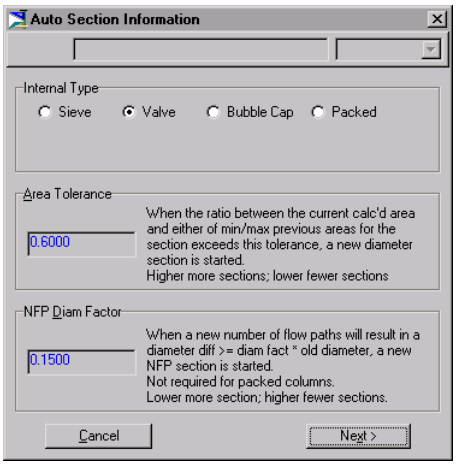
*The **Auto Section** feature creates tower sections of constant diameter based on the parameters you specify.*

The Auto Section feature in HYSYS provides a good starting point for the tray section analysis. In the following steps, you will attach the main tray section of **DeC3** to the utility and use the Auto Section functionality to divide the tower into sections:

1. On the **Setup** page, change the name of the utility, if desired.
2. Press the ***Select TS*** button to access the Object Navigator. Choose **Main TS** for the **DeC3** column and press **OK**.
3. Press the ***Auto Section*** button.

4. Ensure that in the **Internal Type** group, the **Valve** radio button is chosen. Use the default values for **Area Tolerance** and **NFP Diameter Factor**.

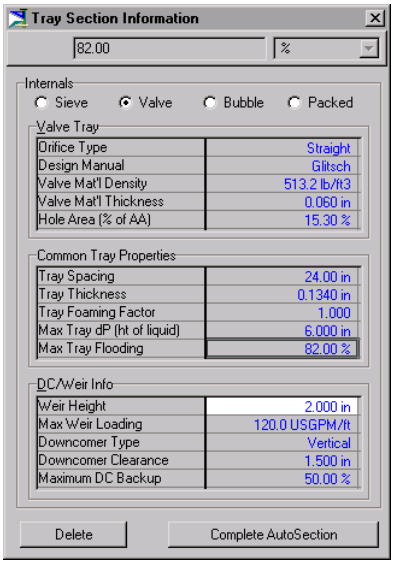
Figure 8.62



5. Once the **Auto Section Information** dialog has been completed as shown in Figure 8.62, press the ***Next*** button and the **Tray Section Information** window appears.
6. In order to **Auto Section**, HYSYS requires the input of design parameters specific to the tower in addition to the selected tower internals. Complete the **Setup** tab as shown in Figure 8.63. Change

the **Tray Thickness** to **0.134 inches**, (10 ga.)and the **Max Tray Flooding** to **82%**. All other parameters can be left at their default values.

Figure 8.63

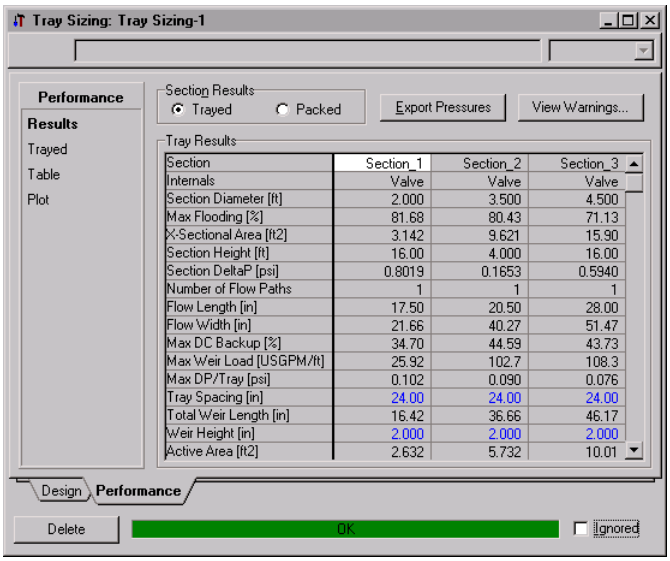


7. Press the ***Complete AutoSection*** button. HYSYS proceeds with the Auto Section calculations. The **Auto Section** view is automatically closed, and you are placed on the **Setup** tab of the main Tray Sizing Utility property view. A summary of the three tray sections calculated is shown on the **Setup** page. The tray sections created are listed below with their corresponding tray ranges.

Tray Section	Tray Range
Section_1	1-8
Section_2	9-10
Section_3	11-18

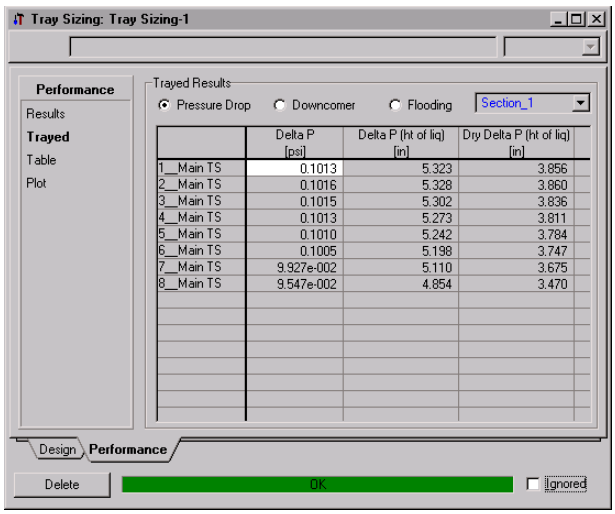
8. Switch to the **Performance** tab for a more detailed description of the generated Auto Section output. It is apparent that the design calculated is well within tolerances.

Figure 8.64



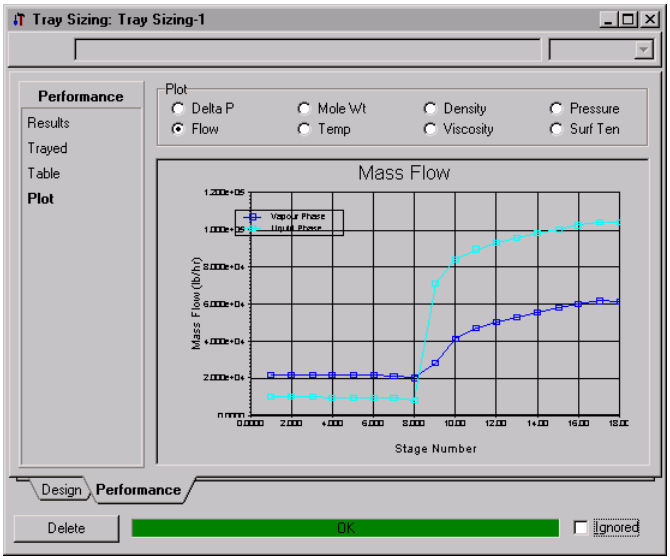
9. On the **Trayed** page, HYSYS displays tray-by-tray information for the highlighted section. By selecting the corresponding radio buttons, information on **Pressure Drop**, **Downcomer**, or **Flooding** can be displayed for each tray section.

Figure 8.65



10. The **Plot** page displays either tables of values or plots for a number of column variables, such as the plot shown in Figure 8.66.

Figure 8.66

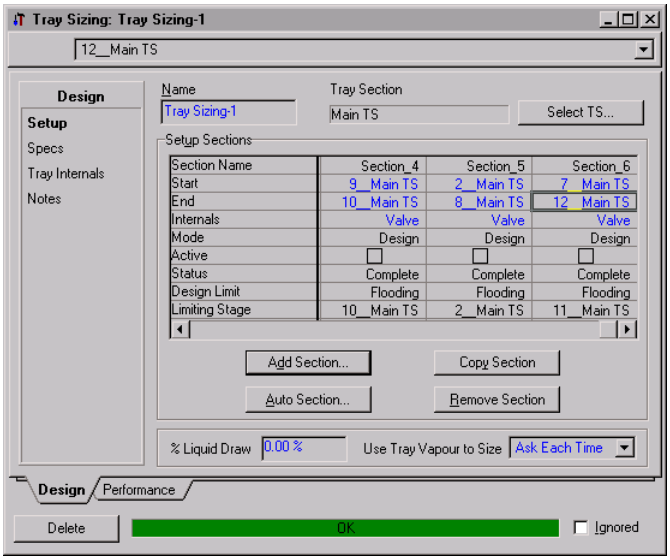


Tray Sizing — Rating Part I

The sections that HYSYS has designed can be manipulated in a rating analysis.

1. To show how close to the design limit a smaller diameter would have been, create a section identical to **Section_2** by returning to the **Setup** page, highlighting **Section_2** and pressing the ***Copy Section*** button. Create two other arbitrary sub-sections by clicking the ***Add Section*** button twice. Specify new tray ranges as shown in Figure 8.67. Note that the tray type (valve) is not changed.

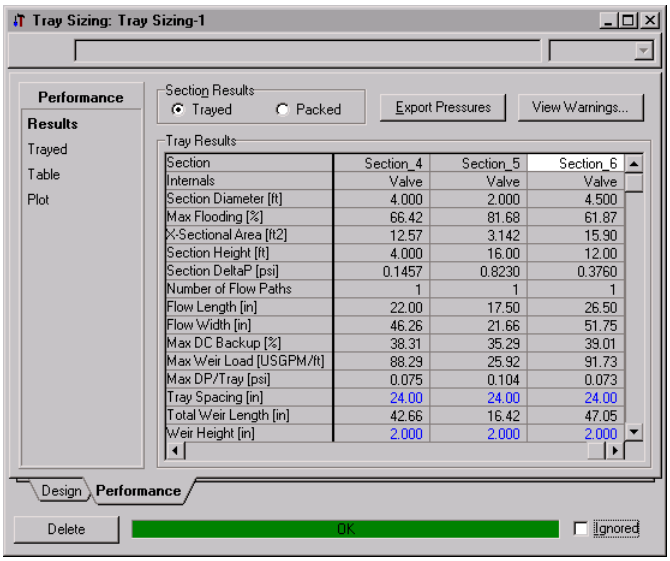
Figure 8.67



2. On the **Specs** page highlight the **Mode** option in **Section_4** and select the **Rating** drop-down option.
3. Enter a value of **1** for the **Number of Flow Paths** and **4 feet** into the **Tray Diameter** cell. HYSYS will automatically calculate the result.

4. Select the **Performance** tab, **Results** page and highlight the cell containing **Section_4**. As seen from Figure 8.68, the weir loading has not exceeded the design parameter of 120 USGPM/ft.

Figure 8.68



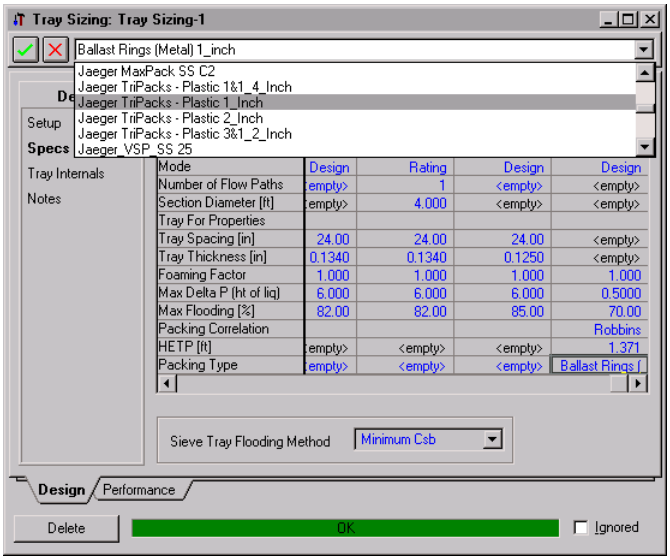
Notice that HYSYS has determined 2.00 and 4.50 foot diameters for **Section_5** and **Section_6**.

8.14.3 Tray Sizing — Rating Part II

Using HYSYS, it is possible to evaluate packed versus trayed columns by changing the internal type from trayed to packed.

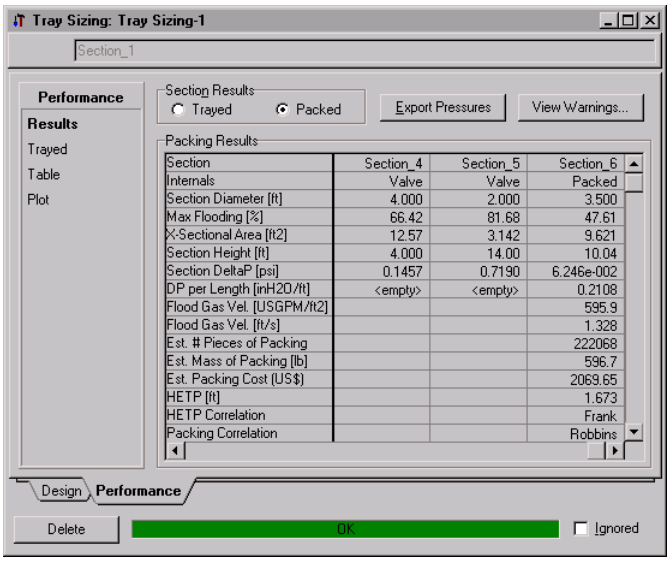
1. Using valve tray internals, the calculated diameter for **Section_6** is 4.50 feet. In order to compare the performance of the same section using Packing, go to the **Specs** page, highlight **Secton_6** and select the **Packed** option from the **Internals** drop-down.
2. Choose *Jaeger TriPacks - Plastic 1_inch* from the **Packing Type** drop down list.

Figure 8.69



3. The section re-calculates automatically.
4. Select the **Results** page and choose the **Packed** radio button from the **Selection Results** group. The Results tab should match **Figure 8.70**.

Figure 8.70



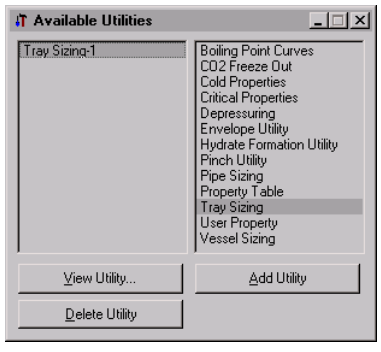
The differences in some of the key parameters when **Section_6** is calculated as a trayed versus a packed section are listed in the table below:

Parameter	Trayed Section	Packed Section
Diameter (ft)	4.5	3.5
Pressure Drop (psi)	0.3760	0.6246
Maximum Flooding (%)	61.87	47.61

8.15 Tray Sizing Manager

The **Tray Sizing** utility is accessed by selecting **Utilities** under **Tools** in the menu bar. Choose the **Tray Sizing** option from the **Available Utilities** view and press the ***Add Utility*** button when it becomes activated.

Figure 8.71



After the **Tray Sizing** utility has been created, the utility's property view is opened. This is the main view used for setting up the utility and accessing the results. This view has two tabs, **Design** and **Performance**, and associated pages, as listed in the following table:

Tab	Page	Description
Design	**Setup**	Manages the column sizing sections.
	Specs	Calculation mode and common tower sizing parameters.
	Tray Internals	Detailed internal specifications.
Performance	**Results**	Overall comparative section results and detailed tray sizing information.
	Trayed	Pressure Drop, Downcomer and Flooding results across the column.
	Table	Tray section physical property profiles in tabular form.
	Plot	Tray section physical property profiles in graphical form.

Note that you can set the default parameters for the Tray Sizing utility from the **Preferences** view (select **Tools**, then **Preferences** in the Menu Bar). On the **Tray Sizing** tab, the defaults for auto section parameters, trayed section and packed section setups can be set.

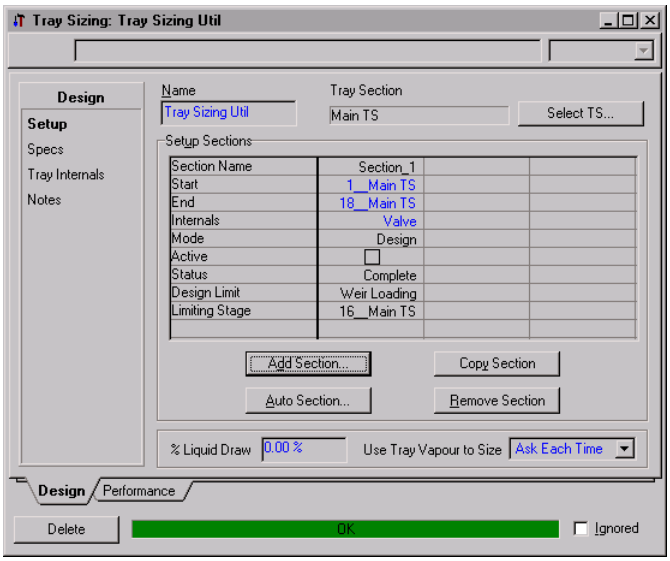
8.15.1 Setup Page

Each **Tray Sizing** utility that is added must correspond to a single column flowsheet tray section. The tray section for which the sizing is desired has to be specified before starting any calculations. Select the ***Select TS*** button located in the top right corner of the tab; select the desired tray section from the resulting column navigator.

*The buttons on the **Setup** page remain greyed-out until a tray section is attached to the utility.*

There are four buttons at the bottom of the matrix on the **Setup** page that allow you to manipulate the number of tray sections attached to your utility.

Figure 8.72



Button	Function
Add Section	Adds a new tray sizing section with the entire span of the target tray section used as the default size, see **Figure 8.72**. This may be changed to a shorter span, if desired, by changing the start and end stages. A preliminary design calculation is automatically performed using all HYSYS default sizing parameters.
Copy Section	You can copy any section already created by selecting the section and then this button.
Auto Section	Automatically calculates the sections in your column. See **Section 8.15.5 - Auto Section** for more information.
Remove Section	Deletes the selected section. Note that HYSYS does not ask for confirmation before removing the section.

HYSYS allows you to create multiple stage sections so that you are able to compare column configurations with different internal types. Therefore, a given span of tray section stages may be sized more than once within a single Tray Sizing utility. However, a give stage may not be included in more than a one active section.

At the bottom of the page there are two fields, the **% Liquid Draw** and **Use Tray Vapour to Size**.

The **% Liquid Draw** allows you to specify the percentage of side liquid draws to be used in the tray sizing calculations. When you specify a liquid percent HYSYS assumes that draw percentage is sitting on the tray. The default value of 0% means that no additional liquid is assumed to be on the tray. So if you enter100%, flooding will increase because you have an additional volume of liquid equal to the draw rate sitting on the tray. This percentage will be equivalent for all trays with draws, you can not specify different percentages for different draws on different trays.

If you have vapour feed(s) attached to your column, HYSYS can size the particular tray to which it is attached either using the vapour feed to the tray section, or the vapour flow leaving it. You can specify which method HYSYS uses with the **Use Tray Vapour to Size** drop-down.

Selection	Calculation Method
Always Yes	HYSYS will use the vapour flow leaving the tray section to size the tray to which a vapour feed is attached for all calculations. The effect of the feed on the tray sizing is considered.
Always No	HYSYS will use the vapour feed to the section to size the tray for all calculations. The effect of the feed is not considered for this method.

Selection	Calculation Method
Ask Each Time	Prior to calculating the Tray Sizing utility, HYSYS will ask you to specify whether to use feed to or vapour flow from the tray as a basis. A message box for each vapour feed to your column will appear prior to the calculations.

In addition to the start and end stages, the following information is available for each section:

- Type of tower internals
- Calculation mode
- Sizing calculation information

Tower Internals

The Tray Sizing Utility provides the option of specifying each section's internal type. Tower internals are broken into four groups:

- Sieve
- Valve
- Packed
- Bubble Cap

Packed Sections

Packed towers are calculated using either Robbins or Sherwood-Leva-Eckert design correlations for predicting pressure drop and liquid hold-up. The tower internals can be selected on the **Specs** and **Tray Internals** pages of the utility property view. You are able to specify the packing type and other parameters specifically related to the packed tower calculations on these pages, as well.

Trayed Sections

The trayed column internals are defined as sieve, valve or bubble cap. Some tray configuration parameters are common to all tray types. There is also a unique set of parameters for each individual tray type. The tower internals can be selected on the **Tray Internals** page of the utility property view.

The calculation methods for the different trays are defined below:

- Valve tray calculations are based on the Glitsch, Koch and Nutter valve tray design manuals.
- Sieve tray calculations are based on the valve tray manuals for tray layout, and Mass-Transfer Operations by Treybal, (McGraw-Hill) for pressure drop, weeping and entrainment calculations.
- Bubble tray calculations are based on the method described in Design of Equilibrium Stage Processes by Bufford D. Smith, (Wiley & Sons).

Section Calculation Modes

*Note that you can only change the **Calculation Mode** on the **Specs** page. On the **Setup** page, the mode is view-only.*

The tray sizing utility has two calculation modes:

- Design
- Rating

Design Mode

In Design mode, HYSYS allows you to perform a design sizing based on the vapour and liquid traffic in the tower. Available design specifications for trayed and packed sections include the type of tower internals, maximum allowable pressure drop, and maximum allowable flooding. For trayed sections the maximum allowable downcomer backup, maximum allowable weir loading, and various other tray parameters may also be specified.

Rating Mode

In Rating mode, HYSYS allows you to perform rating calculations based on a specified tower diameter and fixed tray configuration. If desired, some of the tray dimensions may be left unspecified and HYSYS will automatically calculate design values for them. To perform a rating on a packed section, only the tower diameter is required.

Sizing Calculation Information

There are additional features that provide information on the calculation methods for each tray section:

- Active Check Box
- Status
- Design Limit
- Limiting Stage

Active Check Box

When the **Active** check box is selected for a column section on the **Setup** tab, values calculated in the tray sizing utility are used in the actual column calculations. More than one section may be active in a tray sizing utility. However, the same stage may not be included in more than one active section.

Before updating the column flowsheet with the information from the tray sizing utility, you must change the default arrangement of the pressure profile information in the Column Sub-Flowsheet. On the **Profiles** page of the **Parameters** tab (in the **Column Runner** view), the top and bottom stage pressures must be specified instead of the condenser and reboiler pressures. The condenser and reboiler delta P specifications do not need to be changed. Run the column and then return to the utility.

If your column is in a recycle, there is no automatic update of the pressure profile, it must be done manually.

On the Tray Sizing **Setup** page activate those calculated column sections that you wish to use in your simulation. Proceed to the **Performance** tab, **Results** page and select the ***Export Pressures*** button, which will export the pressure information to the column runner.

Status

This field indicates the status of the tray sizing calculation. The status will read either **Complete** or **Incomplete** on a section by section basis.

Design Limit

This field indicates the design specification that was the last to be satisfied. The five design specifications are:

- Minimum diameter
- Pressure drop
- Flooding
- Weir loading (trayed sections only)
- Downcomer backup (trayed sections only)

This is the critical design specification that is closest to being exceeded if the tower is sized any smaller. For trayed sections, HYSYS uses individual design limits for the required active area and the required downcomer area design calculations.

Limiting Stage

This is the stage in the sizing section on which the design hinges. It is the stage that is closest to exceeding the design specifications.

For trayed towers, there are two limiting trays; one for the tray that was closest to exceeding the design specification while satisfying the section's active area needs, and another one for satisfying the downcomer area.

8.15.2 Design Specifications

The **Specs** page of the utility view allows you to specify the column internals for each section. The following tables outline the available design and tray configuration parameters for the sizing utility.

Design Parameters

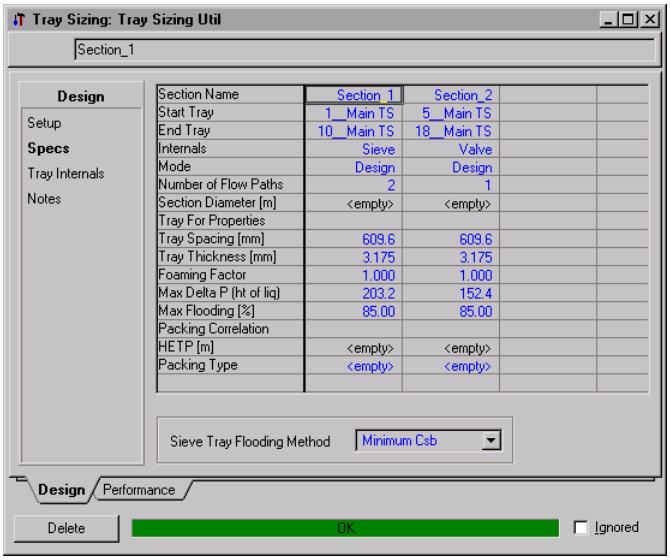
Parameter	Trayed Section	Packed Section
Design Correlation		4
Foaming Factor	4	4
Flooding	4	4
Pressure Drop	4	4
Downcomer Backup	4	
Weir Loading	4	

Tray Configuration Parameters

Parameter	Valve	Sieve	Bubble
Number of Flow Paths	4	4	4
Tray Spacing	4	4	4
Tray Thickness	4	4	4
Weir Height	4	4	4
Downcomer Type	4	4	4
Downcomer Clearance	4	4	4
Design Manual	4		
Hole Area	4		4
Hole Diameter		4	
Hole Spacing		4	
Hole Pitch		4	
Valve Density	4		
Valve Thickness	4		
Orifice Type	4		
Bubble Cap Slot Height			4

8.15.3 Specs Page

Figure 8.73



The **Section Name**, **Start** and **End Trays**, **Internals** and **Mode** displayed on this page are common with the Setup page. However, it is only on the Specs page that you are able to change the calculation mode for the section.

Number of Flow Paths

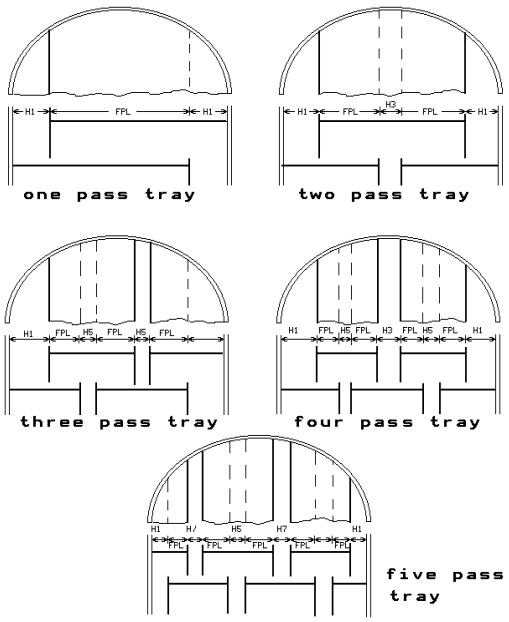
This value represents the number of independent flow paths per tray. Usually a smaller tower diameter can be obtained by using multi-pass trays. However, with more flow paths, there is a reduction in the number of valves or sieve holes that can be placed on the tray This may result in an increase in the pressure drop, an increase in downcomer backup, and a loss in tray efficiency. The following are general guidelines relating the number of flow paths and the tower diameter:

Number of Passes	Min. Diameter (ft)	Pref. Diameter (ft)
2	5	6
3	8	9
4	10	12
5	13	15

If the number of flow paths is not specified, HYSYS will start at one pass and increase the number of passes until the minimum diameter for that number of flow paths is reached. If a smaller number of flow paths is required, a new value may be entered that overrides the calculated NFP. This is accomplished on the **Specs** tab. A new solution will be calculated as soon as the new NFP is entered.

Figure 8.74 summarizes the basic physical layouts of the flow paths available.

Figure 8.74



Tray for Properties

This field is available only in **Rating** mode. You can specify which tray is used to calculate properties for the column.

Tray Spacing

The tray spacing is the vertical distance between trays. Some general guidelines for tray spacing follow:

Expected Tower Diameter (ft)	Suggested tray Spacing (in)
---	12 (minimum)
Up to 4	18 - 20
4 - 10	24
10 - 12	30
12 -24	36

The default value is 24 inches.

Valve and Tray Material Thickness

Since material thickness is often described in terms of gauge, the following table is provided for quick conversions between gauge and inches:

Gauge	Thickness (in)
20	0.037
18	0.050
16	0.060
14	0.074
12	0.104
10	0.134

The default tray thickness is 0.125 inches.

Foaming Factor

This is a measure of the foaming tendency of the system. In general, a lower foaming factor results in a lower overall tray efficiency and requirements for a larger tower diameter.

General Foaming Classification	Foaming Factor
Non Foaming Systems	1.00
Low Foaming Tendencies	0.90
Moderate Foaming Tendencies	0.75
High Foaming Tendencies	0.6

Foaming factors typically seen in some common systems include the following:

Absorbers	Foaming Factor
Ambient Oil (T > 0°F)	0.85
Low Temp Oil (T < 0°F)	0.95
DGA/DEA/MEA Contactor	0.75
Glycol Contactor	0.65
Sulfinol Contactor	1.0

Crude/Vacuum Tower	Foaming Factor
Crude or Vacuum Fractionation	1.00

Fractionators	Foaming Factor
Hydrocarbon	1.00
Low MW Alcohols	1.00
Rich Oil DeC1 or DeC2 (top)	0.85
Rich Oil DeC1 or DeC2 (Btm)	1.0
Refrigerated DeC1 or DeC2 (top)	0.80
Refrigerated DeC1 or DeC2 (btm)	1.00
General Hydrocarbon Distillation	1.00
MEA/DEA Still	0.85
Glycol/DGA Still	0.80
Sulfinol Still	1.00
H_2S Stripper	0.90
Sour Water Stripper	0.50 - 0.70
O_2 Stripper	1.00

Maximum Pressure Drop

The maximum allowable pressure drop per tray may be entered as a height of liquid. If it is not specified, a default maximum of 4 inches of liquid is used. For packed sections, the specification is supplied on a pressure drop per height of packing basis. The default specification is 0.5 inches of water per foot of packing.

Maximum Flooding

The column is sized such that for the given vapour and liquid traffic, the tower flooding will not exceed this specification on any stage. The maximum recommended value is 85% for normal service and 77% for

vacuum or low pressure drop applications. These values will yield approximately 10% entrainment. For diameters under 36 inches a reduced flooding specification of 65 - 75% should be used. A lower value may be specified to allow for contingencies, such as increased capacity. If not specified, a maximum flood factor of 82% is used for flat orifice trays and 77% for venturi orifice trays.

Packed Section Input

Packed tower information can be specified on in the three following parameters. Default values are provided for all packing parameters with the exception of the packing type and the packed section diameter.

Packing Correlation

The Robbins correlation, which is the default selection, is noted to be better at predicting pressure drop and liquid holdup, particularly with newer packing materials. It is valid only at liquid loadings < 20000 lb/hr ft^2. The SLE (Sherwood-Leva-Eckert) correlation should be selected for towers operating above this range.

HETP

The height factor HETP relates packed towers and tray towers. The value refers to the height of packing that is equivalent to a theoretical plate. For design purposes, the most accurate HETP factors are those published by packing manufacturers.

Packing Material

The packing type can be accessed on the **Internals** tab of the utility. A list of the available packing types is shown in the following table.

Packing Type	Material	Packing Type	Material
Ballast Rings	M,P	Jaeger_VSP_SS	M
Ballast Plus Rings	M	Koch-Sulzer(BX) Structured	S
Ballast Saddles	P	Lessing Experimental	M
Berl Saddles	C	Levapacking	P
Cascade MiniRing	M,P,C	Maspak	P
Chempak	M	Montz A-2 Structured	S
Flexipac Mellapac	S	Neo-Kloss Structured	S
Flexirings	M	Norton Intalox Metal Tower Packing	M

Packing Type	Material	Packing Type	Material
Gempak	S	Nutter Rings	M
Glitsch Grid	S	Pall Rings	M,P
Goodloe	S	Protruded	M
Wire Coil Packing	M	Raschig Rings 1/32 in wall	CSteel
Hy-Pak Rings	M	Raschig Rings 1/16 in wall	CSteel
Hyperfil	S	Raschig Rings	C, Carbon
Intalox Saddles	C	Super Intalox Saddles	P
Jaeger MaxPack SS	M	Tellerettes	P
Jaeger Tripacks	P	Cross-Partition Rings	C

The materials used for the different packings (unless otherwise noted) are: Metal(M), Plastic(P), Ceramic(C), and Metal Structured(S).

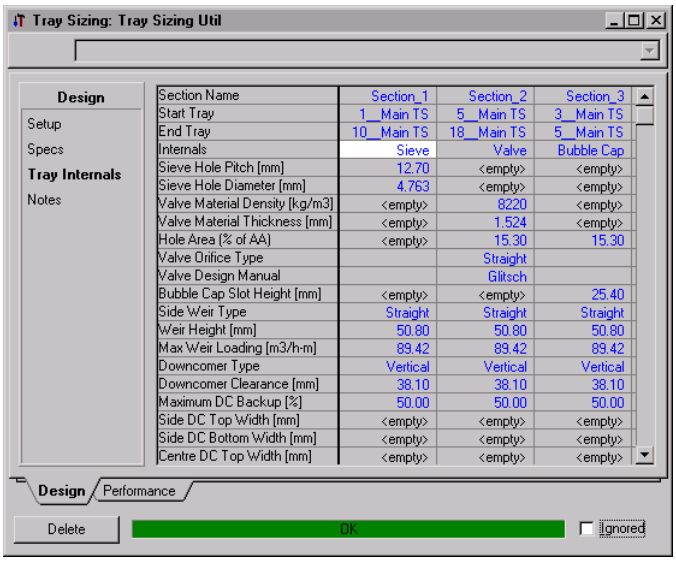
Sieve Tray Flooding Method

The method used to mode sieve tray flooding may be specified using the drop-down box at the bottom of the view. The options are:

- Minimum Csb
- Original Csb
- Fair's Modified Csb

8.15.4 Tray Internals Page

Figure 8.75



If the sizing section is specified as having trayed internals (**Sieve**, **Valve** or **Bubble Cap**), then the internals may be further specified on this page. There are certain column parameters that are common to all trayed columns. You may specify these parameters, or leave them at their default values.

Sieve Tray Parameters

The input available for the configuration of the sieve tray is similar to the valve tray with the exception of the tray input data.

Hole Pitch

Hole pitch refers to the distance between the centers of two adjacent holes. HYSYS requires the hole pitch to be within 1.5 to 5 times the hole diameter. The default hole pitch is 0.5 inches.

Hole Diameter

The default value for the hole diameter is are 0.187 inches.

Valve Tray Parameters

The valve tray is the default tray type for trayed columns in design mode. Defined here are the design parameters specific to valve trays:

Valve Material Density

A table of typical materials used for valves and their associated densities follows:

Valve Material	Density (lb/ft^3)
Carbon Steel	480
Stainless Steel	510
Nickel	553
Monel	550
Titanium	283
Hastelloy	560

Hole Area (% of Active Area)

Note: The Hole Area can also be specified for Bubble Cap trays.

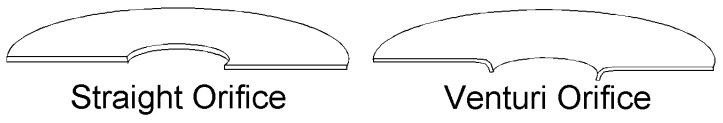
The hole area is the percentage of the active area that is occupied by the valve holes. The default is 15.3%, which corresponds to 12 valves, each having a diameter of 1 17/32 inches, per square foot.

For bubble cap trays, the default hole area is also 15.3%, which corresponds to 12 bubble caps, each having a diameter of 1 17/32 inches, per square foot.

Valve Orifice Type

This is the shape of the hole that is punched in the plate for the valve. As shown in **Figure 8.76**, there are two types of orifices: Venturi and Straight. The straight orifice is used for normal service, while the Venturi orifice is used for low pressure drop applications.

Figure 8.76



Design Manual for Flooding Calculations

Results are presented for flooding calculations from three industry standard design manuals (Glitsch, Koch or Nutter). Any one of the three methods may be selected as the basis for comparison with the maximum allowable % of flood design specification.

The default design manual is Glitsch.

Bubble Cap Trays

Bubble Cap Slot Height

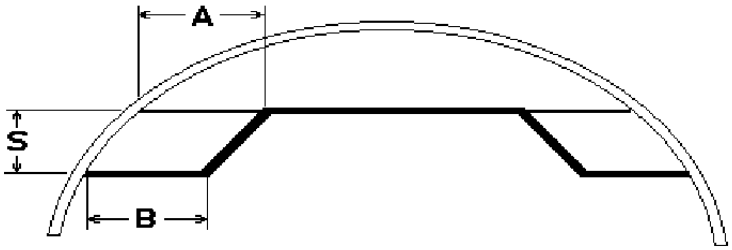
The default slot height is 1.0 inch. This value represents the height of the slots around the base of the bubble caps, through which the gas and liquid are allowed to flow.

Common Tray Parameters

Side Weir Type

This parameter is used to specify the side weir type only. There are two types of weirs available: straight and relief. A relief weir lengthens the side weir without increasing the downcomer area. The relief weir sweeps back, then across the tray, enclosing some active area, as shown in Figure 8.77. A straight weir follows the edge of the downcomer.

Figure 8.77



A relief weir is used for high liquid loads or where a low pressure drop is required, while straight weirs are used for normal service. HYSYS uses a straight weir as the default. However, if the weir loading is above the maximum, a relief weir will be included to alleviate the problem. If a relief weir is installed by HYSYS and a straight weir is desired, a straight weir may be re-specified and the tray rerun in *rating* mode.

Weir Height

The weir height is the distance from the tray to the top of the weir. A weir height of 2 inches is used in most applications. However, a smaller height can be used for low pressure drop or vacuum services. A larger weir height is used to obtain longer residence times (e.g., for chemical reaction services).

The default value for weir height is 2 inches. In general, you can use the following:

Tray Spacing	Weir Height (in)
12	1.5
12 - 24	2
>24	2.5

Maximum Allowable Weir Loading

The weir loading is a measure of the liquid loading on the weirs. Values of 60 - 120 USGPM/ft are typical. Weir loading may be reduced by increasing the number of flow paths or installing a relief weir. A weir loading as high as 240 USGPM/ft can sometimes be tolerated. If the weir loading is not specified, a default value of 96 USGPM/ft is used.

Downcomer Type

There are two types of downcomers available: vertical and sloped. A sloped downcomer has a narrower width at the bottom. This allows more active area and more valves per tray, and also results in a lower pressure drop. Due to cost considerations a vertical downcomer is used for normal service and is the default in HYSYS.

Downcomer Clearance

The downcomer clearance is the distance between the bottom of the downcomer and the tray. The area available for liquid flow under the downcomer is dependent upon this height.

A minimum seal of 0.5 inches is normally recommended. For high liquid velocities and the resulting high pressure drop, this may be reduced. If the downcomer clearance is not specified, a height of 0.5 inches less than the weir height is used. Since the weir height default is 2 inches, this translates to a downcomer clearance default of 1.5 inches.

Maximum Allowable Downcomer Backup

The allowable downcomer backup is measured as the percentage of the tray spacing that the liquid level in the downcomer is allowed to reach. This value represents the average for all the downcomers on the tray. If not specified, a value of 40% is used for services with a vapour density greater than 3 lb/ft^3, 50% for normal densities, and 60% for densities less than 1 lb/ft^3.

The remaining fields on this page are available only when in Rating mode. See the **Section - Column Rating** for more information.

8.15.5 Auto Section

*The Auto Section routine is not available until a tray section has been selected on the **Setup** tab of the tray sizing utility.*

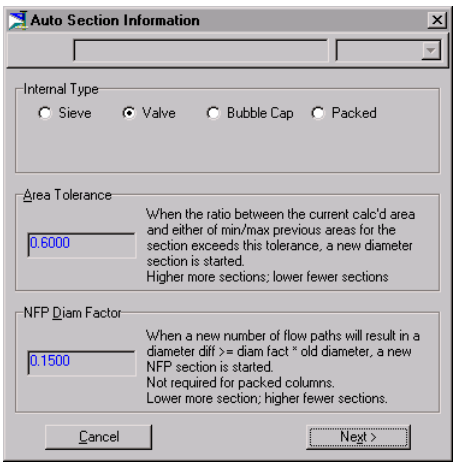
The **Auto Section** function is an optional method in design mode. When using Auto Section, HYSYS automatically calculates the sections for a column. They are then transferred onto the main utility **Setup** tab where you can edit, copy, or delete sections. Auto Section will provide you with an excellent starting point in the design of a tower by performing a summary sizing of the tray section and splitting it into sections of constant diameter, as appropriate.

The sections that HYSYS determines during its analysis are automatically available for you to edit, rename, or delete.

HYSYS allows you to specify the internal type, as well as values for the criteria that are used to start a new section. The two criteria that the user may specify to establish tower sections are:

- Area Tolerance
- NFP Diameter Factor

Figure 8.78



Area Tolerance

The **Area Tolerance** defines the magnitude of change in the calculated area that will cause the start of a new section. HYSYS first performs a design for *stage i*, using the current parameters for the chosen internals, and the NFP for the current section (valve, sieve, and bubble trays only) to determine the required area. This area is compared to the minimum and maximum areas for the current section, which HYSYS retains.

If the magnitude of difference for either comparison exceeds the **Area Tolerance**, a new section is started beginning at *stage i*. The previous section will be assigned the maximum diameter for that section. If the calculated area for *stage i* is outside the range defined by the minimum and maximum for the section but does not exceed the tolerance, the calculated area for *stage i* will replace the appropriate stored value.

NFP Factor

When the comparison of areas is complete, HYSYS will recalculate the required area of each stage using a different Number of Flow Paths. This area is compared with the previously calculated area for each stage. If the magnitude of the change is greater than the **NFP Diam Factor**, a new section is started.

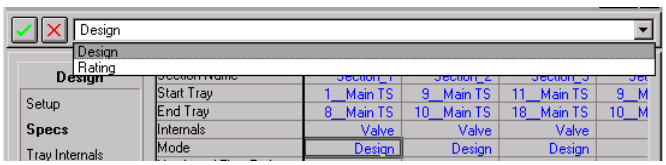
The entire column is stepped off in the above manner according to the Area and NFP guidelines and sections of constant diameter, number of flow paths, active area, and downcomer area are defined for the tower. When this initial sizing is complete, HYSYS re-rates each tray based on a diameter calculated from the maximum downcomer and active area required for trays in that section. This value is available on the **Results** tab of the utility property view. The limiting factor(s) for each section are displayed on the **Setup** tab.

Using **Auto Section**, you are required to specify the tray internal type, either Sieve, Valve, Bubble Cap or Packed. In addition, you have the option of specifying the parameters for the chosen configuration. If no parameters are specified, HYSYS uses values from its default set.

Column Rating

Column sections may be rated using the **Rating** Calculation Mode on the **Specs** page. Select Rating from the drop-down list in the title bar, as shown in Figure 8.79.

Figure 8.79



If you modify the Main Flowsheet or Column Sub-Flowsheet, the tray sizing utility redesigns and rerates all of the sections based on the current configuration using the new stage by stage traffic, physical and transport property information from the column Sub-Flowsheet.

Trayed Section Rating

For trayed sections, the minimum required information for HYSYS to calculate the section performance includes the **number of flow paths** and the **column diameter**.

If desired, you can supply the following downcomer widths on the **Tray Internals** page:

- Side top and optional bottom
- Centre top and optional bottom
- Off-side top and optional bottom
- Off-centre top and optional bottom

The optional bottom width allows for the specification of sloped downcomers. Downcomer widths that are not supplied will be calculated at optimal design values for the given number of flow paths.

The remaining tray configuration parameters can be specified as discussed in **Section 8.15.2 - Design Specifications**. Once rating parameters have been set, the ***Calculate*** button on the **Specs** tab must be pressed before HYSYS will complete the rating calculations.

Packed Section Rating

For packed sections, the required information for HYSYS to calculate the section performance includes both the **Section Diameter** and the **Tray for Properties** on the **Tray Internals** page.

The remaining tray configuration parameters can be specified as discussed in **Section 8.15.2 - Design Specifications**. Once rating parameters have been set, HYSYS will complete the rating calculations. If HYSYS is unable to complete the calculations, this will be indicated on the property view status bar. To view the warnings generated, go to the **Results** page of the **Performance** tab and select the ***View Warnings*** tab.

8.16 User Properties

*Refer to Chapter 6 - User Properties of the **Simulation Basis** manual for detailed information concerning User Properties.*

The User Property utility allows you to define new properties based on the composition of a stream. You select the Mixing Rule formula as well as the Mixing Basis. Each component in the stream has its own property value specified and the total property value is calculated for the current composition of the stream. Possible uses of the User Property include as a specification in a distillation column or as a target variable in an **ADJUST** operation.

8.16.1 Quick Start

To illustrate one application of the User Property utility, part of a simple refrigeration gas plant is simulated. The Higher Heating Value will be installed as a user property.

Install a Fluid Package using the **Peng Robinson** property method and the required components of the **FEED** stream. Define the stream **FEED**, which will be used as the main feed for the case.

MATERIAL STREAM [FEED]		
Tab [Page]	**Input Area**	**Entry**
Worksheet [Conditions]	Temperature	60.0000 °F
	Pressure	600.0000 psi
	Molar Flow	144.0000 lbmole/hr
Worksheet [Composition]	C1 Mole Frac	0.48611
	C2 Mole Frac	0.13889
	C3 Mole Frac	0.06944
	i-C4 Mole Frac	0.05556
	n-C4 Mole Frac	0.06250
	i-C5 Mole Frac	0.04167
	n-C5 Mole Frac	0.04861
	n-C6 Mole Frac	0.04861
	n-C7 Mole Frac	0.02778
	n-C8 Mole Frac	0.02083

Install the following unit operations and logicals.

SEPARATOR [SEP]		
Tab [Page]	**Input Area**	**Entry**
Design [Connections]	Inlet	FEED
	Vapour Outlet	SEPVAP
	Liquid Outlet	SEPLIQ
Design [Parameters]	Pressure Drop	0 psi

SEPARATOR [LTS SEP]		
Tab [Page]	**Input Area**	**Entry**
Design [Connections]	Inlet	COLDGAS
	Vapour Outlet	LTSVAP
	Liquid Outlet	LTSLIQ
Design [Parameters]	Pressure Drop	0 psi

HEAT EXCHANGER [GAS-GAS]		
Tab [Page]	**Input Area**	**Entry**
Design [Connections]	Tube Side Inlet	SEPVAP
	Tube Side Outlet	COOLGAS
	Shell Side Inlet	LTSVAP
	Shell Side Outlet	SALESGAS
Design [Parameters]	Tubeside Delta P	10 psi
	Shellside Delta P	10 psi

HEAT EXCHANGER [CHILLER]		
Tab [Page]	**Input Area**	**Entry**
Design [Connections]	Tube Side Inlet	COOLGAS
	Tube Side Outlet	COLDGAS
	Shell Side Inlet	C3-3
	Shell Side Outlet	C3-4
Design [Parameters]	Tubeside Delta P	10 psi
	Shellside Delta P	0.1 psi
Work Sheet [Conditions]	COLDGAS Temperature	0 °F

BALANCE [DP]		
Tab [Page]	**Input Area**	**Entry**
Connections	Inlet Streams	SALESGAS
	Outlet Streams	DEWPT
	Auto Calculation	Check this box
Parameters	Balance Type	Mole
Work Sheet	DEWPT Vapour Fraction	1
	DEWPT Pressure	815 psi

ADJUST [LTSTEMP]		
Tab [Page]	**Input Area**	**Entry**
Connections	Adjusted Variable	COLDGAS Temperature
	Target Variable	DEWPT Temperature
	Spec. Target Value	14.0 °F
Parameters	Method	Secant
	Tolerance	0.001 °F
	Step Size	2 °F
	Minimum	-4 °F
	Maximum	20 °F
	Maximum Iterations	10

SET [DEL T]		
Tab [Page]	**Input Area**	**Entry**
Connections	Target Y	SALESGAS Temperature
	Source X	SEPVAP
Parameters	Mulitplyer	1
	Offset	10

Access the User Property utility by selecting **Utilities** under **Tools** in the Menu Bar. Choose **User Property** from the list and press the ***Add Utility*** button.

Generalized Procedure

1. On the **Connections** tab, change the name of the utility, if desired.
2. Press the ***Select Stream*** button and choose a stream from the Object Navigator.
3. Press the ***Add Property*** button.

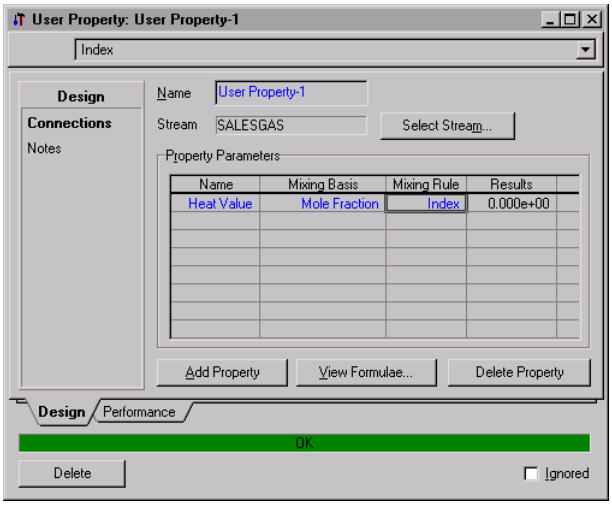
*Refer to the **Chapter 6 - User Properties** of the **Simulation Basis** guide for explanations of the various terms.*

*Press the **View Formulae** button to view the selected Mixing Rule formula.*

4. You can now change the property parameters:
   - Name
   - Mixing Basis (Fraction or Flow): Mole, Mass, Liquid Volume
   - Mixing Rule: Algebraic, Log, Index.
5. Repeat steps #3 and #4 until all properties have been defined for the stream.
6. Switch to the **Prp. Values** tab.
7. Highlight the User Property from the **Selected Property** group box.
8. Input the property values for each component in the **Property Values** group box.
9. Modify the two formula parameters to more accurately reflect your property formula.
10. Repeat steps #7 through #9 for each User Property.
11. Switch back to the **Connections** tab to view the current value of the User Property in the **Property Results** cell.

Design Tab

Figure 8.80



A Higher Heating Value for the product stream **SALESGAS** should be installed as the User Property. Press the ***Select Stream*** button and choose the stream from the Object Navigator.

Press the ***Add Property*** button. Complete the **Connections** page as shown. The Mole Fraction basis is used in the Algebraic Mixing Rule

equation. You can view the equation structure by pressing the ***View Formulae*** button. The algebraic equation has the following format:

$$(P_{mix})^{F1} = F2 \times \sum_{i=1}^{N} (x(i)P(i)^{F1}) \qquad (8.15)$$

P_{mix} - total user property value

P(i) - property value for component

x(i) - component fraction

F1, F2 - specifiable equation parameters

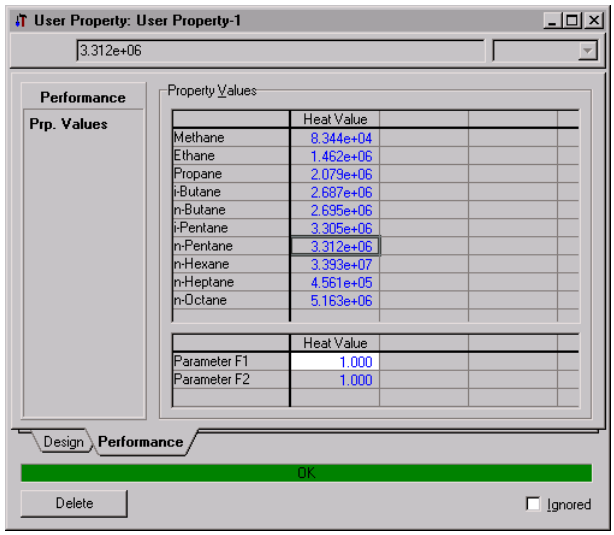
The **Property Results** cell will show a value of 0.0000 until the user property has been defined.

Results
9.862e+05

Property Values Tab

On the **Performance** tab (**Prp. Values** page), enter the component property values. Leave the mixing rule equation parameters at their default values. The total property value can be viewed on the **Connections** tab. The SALESGAS stream has a Higher Heating Value of 9.862E+05.

Figure 8.81



8.17 Vessel Sizing

For a comprehensive costing and sizing software package for your entire case, Economix is available. See your nearest office or agent for details.

The Vessel Sizing utility allows you to size and cost installed separator, tank and reactor unit operations. You can select a vertical or horizontal orientation for the separator. To obtain a more effective analysis for your vessel, changes can be made to the default parameters provided by HYSYS.

8.17.1 Quick Start

Prior to performing the cost analysis of the separator, the vessel and its attached streams must be installed. First, install a Fluid Package using the **Peng Robinson** property method and the required components for the **Feed** stream.

Create the stream **Feed** with the following specifications:

MATERIAL STREAM [Feed]		
Tab [Page]	**Input Area**	**Entry**
Worksheet [Conditions]	Temperature	60.0000 °F
	Pressure	600.0000 psi
	Molar Flow	5000.0000 lbmole/hr
Worksheet [Composition]	Methane Mole Frac	0.4861
	Ethane Mole Frac	0.1389
	Propane Mole Frac	0.0694
	i-Butane Mole Frac	0.0625
	n-Butane Mole Frac	0.0556
	i-Pentane Mole Frac	0.0486
	n-Pentane Mole Frac	0.0417
	n-Hexane Mole Frac	0.0486
	n-Heptane Mole Frac	0.0278
	n-Octane Mole Frac	0.0208

SEPARATOR [Sep]		
Tab [Page]	**Input Area**	**Entry**
Design [Connections]	Inlet	Feed
	Vapour Outlet	SepVap
	Liquid Outlet	SepLiq
Design [Parameters]	Pressure drop	0 psi
	Duty	0 kJ/hr

To begin the vessel sizing analysis, choose the **Utilities** option under **Tools** in the Menu Bar. Highlight **Vessel Sizing** and press the ***Add Utility*** button.

Generalized Procedure

1. On the **Connections** page, change the name of the utility, if desired.
2. Press the ***Select Separator*** button and choose the vessel for the analysis from the Object Navigator.

*Pressing the **Set Defaults** button will return **ALL** of the original data provided by HYSYS.*

3. Select a radio button for the **Orientation** of the vessel: **Horizontal** or **Vertical**.
4. On the **Sizing** page, select the following specifications from the **Available Sizing Specifications** group. Specifications can be added or removed from the **Active Sizing Specifications** using the ***Add Specs*** or ***Remove Specs*** button, respectively. The following specifications are particular to the orientation of the separator:
   - Max. Vapour Velocity
   - Diameter
   - L/D Ratio
   - Vapour Space Height
   - Demister Thickness
   - Liquid Residence Time
   - Liquid Surge Height
   - Total Length - Height
   - Nozzle to Demister
   - Demister to Top
   - LLSD (Low Level Shut Down)
   - Total Separator Height
5. On the **Construction** page, specify any of the following information:
   - Chemical Engineering Index
   - Material Type: Carbon Steel, SS 304, SS 316, Aluminium
   - Mass Density
   - FMC (material of fabrication factor)
   - Allowable Stress
   - Shell Thickness
   - Corrosion Allowance

*HYSYS recalculates after each change is made on the **Parameters** tab or on the **Construction** tab.*

*View the various equations by pressing the **Equation Help** button.*

6. Modify the factors used in the sizing and cost equations on the **Cost Parameters** tab. These factors deal with the **Base Cost, Shell Thickness, Accessories Cost** and **Shell Mass**.

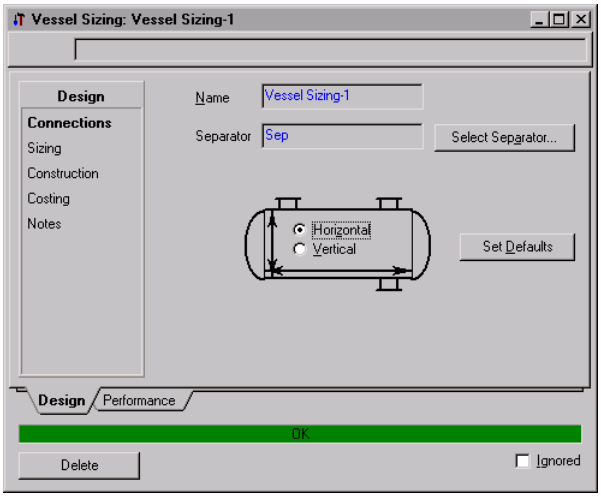
7. View the results of the cost analysis on the **Costing** tab. The **Base Cost**, **Associated Cost** and **Total Cost** are listed in $US.
8. A summary of the **Vessel Results** is provided on the **Results** tab.

Cost Analysis - Horizontal vs. Vertical

Horizontal

On the **Connections** page, press the ***Select Separator*** button. From the Object Navigator, select SEP as the vessel for the analysis.

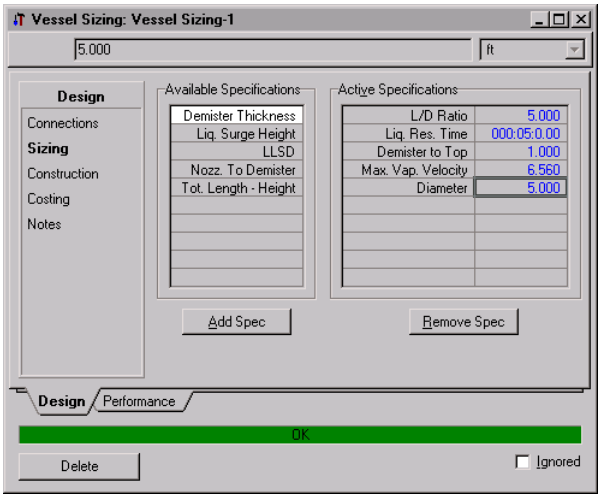
Figure 8.82



Complete the **Sizing** page and **Construction** page, as shown in Figure 8.83 and Figure 8.84, for the Horizontal separator. Select the **Max. Vap. Velocity** specification from the **Available Sizing Specifications** group and press the ***Add Spec*** button. Set the value as **6.56** ft/s. Change the default **L/D Ratio** value to **5**.Move the **Diameter** specification to the **Active Sizing Specification** group and specify a **5** ft diameter for the vessel.

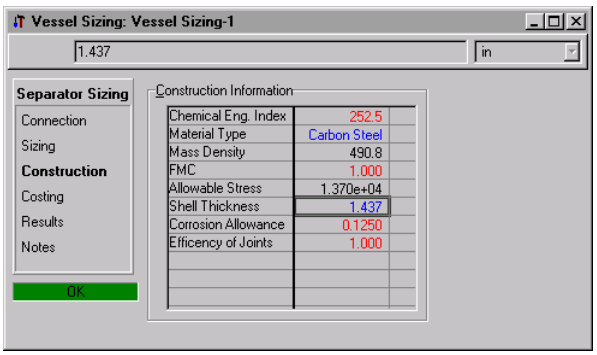
> If the diameter is not specified, HYSYS will change the value when the vessel orientation is switched.

Figure 8.83



The separator is constructed from carbon steel. Change the **Shell Thickness**. The **Shell Mass** will be calculated as 29141.65 lbs.

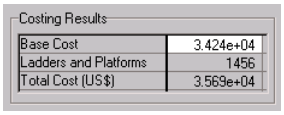
Figure 8.84



All of the **Cost Parameters** will be left at their default values.

On the **Costing** page, the **Base Cost** for the separator is shown. The **Associated Cost** incorporates the amount required for the purchase of additional equipment, such as ladders and platforms. All dollar values are in $US. HYSYS determines the total cost of a horizontal orientation of **SEP** to be $35690.00.

Figure 8.85

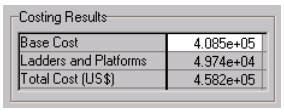


Vertical

To examine what the resulting design would be for a vertical orientation, select the **Vertical** radio button on the **Parameters** page. For this preliminary comparison between the horizontal and vertical orientations simply change the orientation to **Vertical**. HYSYS will then produce a sizing and costing analysis.

As can be seen on the **Costing** page, the vertical separator is more expensive than the horizontal separator. Although the calculated shell weights are virtually the same, the cost of producing a vertical separator is significantly higher.

Figure 8.86



8.18 Adjust Operation with Utility Target Object

A powerful feature of the Utilities is the ability to incorporate them in the **ADJUST** Operation. For example, you may wish to determine the amount of Inhibitor that should be mixed with a Feed Stream (see Section 8.9 - Hydrate Formation) such that the hydrate formation temperature is reduced to -40°C. The streams are defined as follows, using the **Peng Robinson** property method:

MATERIAL STREAMS [FEED, INHIBITOR]			
Tab [Page]	**Input Area**	**FEED Entry**	**INHIBITOR Entry**
Worksheet [Conditions]	Temperature	-5.0000 °C	-5.0000 °C
	Pressure	7000.0000 kPa	7000.0000 kPa
	Molar Flow	50.0000 kgmole/hr	0.0000 kgmole/hr
Worksheet [Composition]	Nitrogen Mole Frac	0.0590	0.0000
	Methane Mole Frac	0.7091	0.0000
	Ethane Mole Frac	0.0468	0.0000
	Propane Mole Frac	0.0192	0.0000
	n-Butane Mole Frac	0.0078	0.0000
	n-Pentane Mole Frac	0.0078	0.0000
	CO2 Mole Frac	0.1404	0.0000
	H2O Mole Frac	0.0099	0.2000
	Methanol Mole Frac	0.0000	0.8000

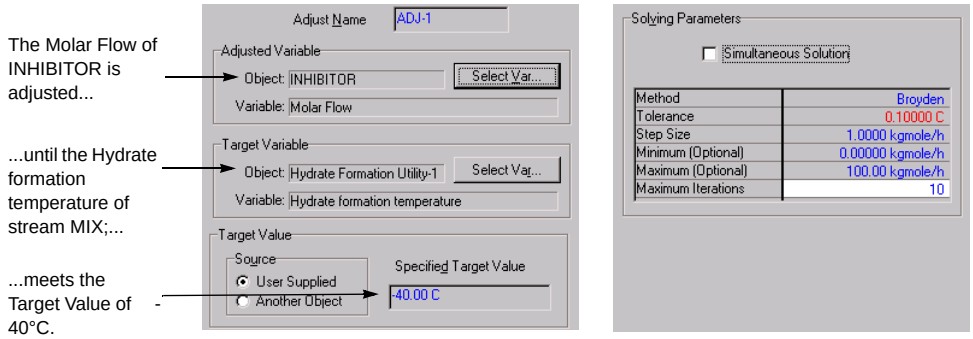
Note that the flow rate of stream **INHIBITOR** is initially set to 0 kgmole/hr.

First, you will need to set up a **MIXER** operation to combine streams **FEED** and **INHIBITOR**, and attach the Hydrate Utility to stream **MIX**.

MIXER [MIX-100]		
Tab [Page]	**Input Area**	**Entry**
Connections	Inlets	FEED INHIBITOR
	Outlet	MIX

Finally, set up the **ADJUST** Operation such that the molar flowrate of stream **INHIBITOR** will be adjusted until the hydrate formation temperature of stream **MIX** is reduced to -40°C.

Figure 8.87



When the **ADJUST** has solved, you can view the results in the Workbook. To meet a Hydrate formation temperature of -40°C, the molar flow of **INHIBITOR** has been increased to 1.219 kgmole/hr.

*For more information on the Adjust operation, see **Section 10.1 - Adjust** of the **Steady State Modelling** manual.*

Figure 8.88

Name	FEED	INHIBITOR	MIX
Vapour Fraction	0.9901	0.0000	0.9641
Temperature [C]	-5.000	-5.000	-4.603
Pressure [kPa]	7000.	7000.	7000.
Molar Flow [kgmole/h]	50.00	1.219	51.22
Mass Flow [kg/h]	1133.	35.64	1168.
Liquid Volume Flow [m3/h]	2.751	0.04367	2.794
Heat Flow [kJ/h]	-6.133e+06	-3.100e+05	-6.443e+06

8.19 References

[1] van der Waals, J.H., Platteuw, J.C., Advan Chem Phys, 2, 1 (1959).

[2] Parrish, W.R., Prausnitz, J.M., I.E.C. Proc Des Dev, 11, 26 (1972).

[3] Sload, E.D., Khoury, F., Kobayashi, R., I.E.C. Fundam, 15, 318 (1976).

[4] Ng, H.J., Robinson, D.B., Ind Eng Chem Fundam, 19, 33 (1980).

[5] Marsland, R.H., "A User Guide on Process Integration for the Efficient Use of Energy", Insitution of Chemical Engineers, England, 1982.

Index

A

Active View 1-15
Adding Components
    See Basis Environment.
Adjust Operation
    utility target object 8-118
Annotations 3-52
    changing font and colours 3-54
    editing in PFD 3-52, 3-53
    moving and sizing labels 3-53
Area Tolerance
    See Column Sizing Utility.
Auto Section.
    See Column Sizing Utility.
Axes
    formatting 6-20

B

Basis Environment
    entering 7-15
    selecting components 1-32
Basis Manager 1-31
    adding fluid packages 1-31
    menu bar commands 7-59
Boiling Point Curve Utility 8-4
    cold properties 8-8
    critical properties 8-7
    example 8-6
    plots 8-8
Boiling Ranges 7-16
Boundary Label
    See Flowsheet.
Button Bar 1-12

C

Calculation/Responsiveness Button 1-11
Case
    Snap Shot 5-33
    states 5-33
Case Studies
    3-dimensional graph control 5-44
    adding 5-38
    display properties 5-41
    multi-dimensional graphing 5-43
    removing 5-39
    setup 5-39
Closing a Case 7-8
Closing Views 1-29
Cold Properties Utility 8-12
Colour Schemes 3-30
    setting preferences 7-50
Column
    accessing PFDs 3-23
    disabling input experts 7-34
    menu 7-59
    packed section rating 8-108
    PFD 3-13
    sizing utility 8-79
    subflowsheet environment 2-6
    tray section display in PFD 3-39
    trayed section rating 8-108
Column Sizing Utility 8-79
    area tolerance 8-106
    auto section 8-106
    calculation information 8-93
    calculation modes 8-93
    design 8-81
    design specifications 8-95
    generic tower parameters 8-96
    material thickness 8-104
    NFP factor 8-107
    packed section input 8-100
    packed sections 8-92
    quick start 8-69, 8-79
    tray spacing 8-98
    Trayed Sections 8-92
    trayed sections 8-92

Component Maps 2-19
Confirm Mode Switches 7-34
Copy Command 7-9
Copy Stream Conditions
See Streams.
Critical Properties
See Utilities.
Critical Properties Utility
analysis 8-15
quick start 8-14
true and pseudo 8-15
Cursors
changing default 7-54
Cut Command 7-9

D

Data Historian.
See Strip Charts.
Data Recorder
deleting a scenario 5-36
deleting states 5-36
setup 5-36
DataBook 5-15
adding scenarios 5-33
case studies tab 5-37
data recorder tab 5-33
process data tables tab 5-17
strip charts tab 5-20
variables tab 5-16
DCS 7-29
Decanter Example.
See Flowsheet.
Deleting
confirm mode 7-34
PFDs 3-16
Dependent Variables
See Property Table Utility.
Depressuring Utility 8-17
compressor loop system 8-37
connections 8-25
heat flux 8-28
plots 8-36
quick start 1 8-18
quick start 2 8-22
types 8-17
valve equations 8-32
vessels 8-35
Depressuring Utiliy
adiabatic mode 8-30
fire mode 8-29
mode 8-27
relief pressure 8-27
DeskTop 1-11
button bar 1-12
main features 1-11
Desktop Preferences 7-35
Desktop Setup
See Preferences.
Drop Down Lists 1-26
During 7-16
DXF Files 6-16
Dynamic Preferences 7-38

E

Edit Menu
cut/ copy/ paste commands 7-9
Enter Basis Environment 7-15
Envelope Utility 8-40
connections 8-41
PF-PH-PS 8-43
plots 8-42
pressure-temperature 8-42
TV-TH-TS 8-44
Environments 2-4
advantages 2-9
column subflowsheet 2-6
entering build 5-10
main flowsheet 2-6
oil characterization 2-5
relations 2-7
simulation 2-5
simulation basis 2-4
subflowsheet 2-6
Event Scheduler 7-14
Export Flowsheet Button 7-11, 7-13
Extensions 7-56

F

Face Plates
menu bar commands 7-28
opening 3-25
preferences 7-35
File Menu 1-30, 7-4
closing a case 7-8
save 7-7
starting a simulation 7-4
Files
Hyprotech file picker 7-48

saving locations 7-49
Files Preferences 7-48
Fire Mode
See Depressuring Utility.
Flowsheet
adding utilities 5-50
automatic naming of objects 7-36
blank flowsheets 2-22
boundary label 2-34
column 2-3
decanter example 2-23
feed and product connections 2-16
information transfer 2-14
multi-level architecture 2-12
navigation 5-8
object browser 5-7
parameters 2-17
special elements 1-10
tags 2-15
transfer basis 2-34
user variables 7-18
Flowsheet Analysis
using the PFD 3-19
Flowsheet Menu
boiling ranges property view 7-16
fluid packages 7-21
reaction package 7-18
user properties 7-21
Flowsheet Object Browser 5-7
Fluid Packages 7-21
in subflowsheets 2-11
Fly-by Information Boxes 3-21
Flybys 7-37
Fonts
changing default 7-52
Format Editor 6-23
Format Preferences 7-44

G

Graph Control 6-18
axes tab 6-20
data page tab 6-19
legend tab 6-22
plot area tab 6-23
title tab 6-21

H

Help Options 7-62
Home View
preferences 7-35
Hot Keys 1-20
Hydrate Formation Utility 8-45
formation pressure 8-49
formation temperature 8-49
hydrate inhibition 8-50
hydrate inhibition example 8-51
stream conditions 8-47
stream settings 8-45
HYSIM
case limitations 7-6
HYSIM Case
opening in HYSYS 7-5
reading 7-5

I

Icons
changing default preferences 7-53
changing icons on PFD 3-43
wire frame/3D 3-44
Information Transfer 2-14
Input
Editing 1-28
numerical 1-25
supplying 1-24
Text 1-24
Input Experts Option 7-34
Installing
templates 2-37
Installing Operations
with property view access 1-45
without property view access 1-46
Installing Streams
with property view access 1-35
without property view access 1-36
Integrator 7-14
Interface
elements 1-4
flowsheet elements 1-10
objects 1-6
Interface Basics 1-4
Interface Terminology 1-13

L

Legend 6-22

M

Macro Language Editor 7-32
Mapping 2-19

Menu Bar
access 7-3
basis 7-59
closing a case 7-8
column 7-59
edit 7-9
file 7-4
flowsheet 7-17
help 7-62
PFD 3-14, 7-26
printing 6-4
saving 7-7
simulation 7-10
simulation navigator 7-18
starting a simulation 7-4
tools 7-27
window 7-60
workbook 7-25
Modal Views 1-18
Moving
between views 1-22
through a view 1-21
To Up/Downstream Object 1-23
Moving to the Up/Downstream Object 1-23
Multi-Flowsheet Architecture 1-5

N

Naming Preferences 7-36
Navigation 5-8
between flowsheets 2-14
Neueral Networks *See Parametric Utility.*
NFP Factor
See Column Sizing Utility.
Non-Modal Views 1-18

O

Object Browser 5-7
Object Deselection 3-36
Object Filter 5-9
Object Inspection 3-4
Object Moving 3-36
Object Navigator 1-9, 5-9
multi-flowsheet navigation 2-14
Object Palette 1-33
installing operations 1-34
installing streams 1-35
opening and closing 7-18
Object Selection
multiple in PFD 3-35
single in PFD 3-34
Object Status Window 5-46
object inspection 5-47
Objects 1-6
finding 7-18
optimization 7-18
selecting for utility 5-10
Oil Characterization Environment
See Environments.
Oil Output Settings 7-16
Oils Preferences 7-57
Operations
analysis 5-5
connecting in PFD 3-12
deleting from workbook 4-10
deletion 1-51
installation 1-44, 7-17
installation example 1-47
installing from object palette 1-34
installing from workbook 4-5
performance 5-5
subflowsheet 2-10
Optimizer 7-13

P

Package Properties 7-57
Packed Towers
See Column Sizing Utility.
Pan/Zoom Functions.
See PFD.
Parametric Utility 8-52
Paste Command 7-9
Performance Page 7-39
PFD
adding annotations 3-52
aligning objects 3-37
auto positioning 3-36
auto-scrolling 3-45
break connection button 3-7, 3-14
cloning objects 3-19
colour schemes 7-50
column tables 3-30
connecting logical operations 3-13
connecting operations to streams 3-11
connecting streams and operations 3-10
connecting two operations 3-12
creating new objects 3-11
Cut/Copy/Paste 3-19
Cut/Paste Functions 3-17

deleting streams and operations 3-9
disconnecting streams and operations 3-13
Exporting Objects 3-18
hiding objects 3-54
Importing Objects 3-18
installing streams and operations 3-9
label variables 3-21
line segments
adding bend points 3-48
alignment 3-51
line straightening 3-50
moving 3-48
removng bend points 3-50
manipulating objects 3-34
menu bar options 7-26
modes 3-6
multi-pane 3-55
object transformation 3-42
object variable table 3-26
open PFD option 3-24
printing 6-7
printing as a DFX file 6-16
quick route 3-8
rebuilding 3-52
rerouting streams 3-46
show/hide sub-flowsheet objects 3-24
sizing objects 3-38
stream label options 3-21
swap connections 3-41
swap connections button 3-7
table properties 3-26, 3-28
tables 3-26
thick stream line 3-45
tools 3-5
view 1-8
PFD Colour Schemes 3-30
adding query colour scheme 3-32
changing 3-31
deleting query colour scheme 3-33
editing query colour scheme 3-33
selecting 3-31
PFD Modes
attach 3-6
move 3-6
size 3-6
PFD Notebook 3-15
deleting a PFD 3-16
installing a new PFD 3-15
renaming a PFD 3-16
Pinch Utility
quick start 8-65
results 8-67
Pipe Sizing Utility 8-69
Playback
See Script Manager.
Plots
printing 6-8
Plottable Properties 7-58
Point Properties 7-58
Preference Set
colours 7-50
cursors 7-54
fonts 7-51
icons 7-53
sounds 7-56
Preferences
extenstions tab 7-56
oil input 7-57
reports tab 7-45
resources tab 7-50
simulation tab 7-33
stream configuration 7-57
tray sizing 7-57
variables tab 7-40
Prererences
files tab 7-48
Print Spec Flowsheet Tab 7-11, 7-13
Printer Setup 6-9
Printing
menu bar command 7-8
operation specsheets 6-6
PFD 3-57, 6-7
plots 6-8
snap shots 6-3, 6-9
stream specsheets 6-7
workbook specsheets 6-5
Printing Options 6-4
Property Table Utilites
independent variables 8-75
Property Table Utility 8-73
dependent variables 8-76
plots 8-76
quick start 8-73
tables 8-77
Property View
accessing 1-16
flowsheet analysis 5-3
locating 5-10

pages 1-17
tabs 1-17
Property Views
cells 1-17
highlight location 1-18
modal vs. non-modal 1-18

Q

Quick Route 3-8

R

Reaction Package 7-18
adding 7-19
opening 7-18
Real Format Editor
See Format Editor.
Recording
See Script Manager.
Report Builder
contents tab 6-11
page setup tab 6-14
Report Manager 6-10
Report Preferences 7-45
Reports 6-10
company info 7-47
creating 6-11
datasheets 7-46
deleting 6-15
editing 6-14
format/layout 7-45
printing 6-15
Resource Preferences 7-50

S

Saving
file locations 7-49
save as 7-7
workspace 7-60
Saving a Case
save all 7-7
Saving a Simulation 7-7
Scenarios
adding to databook 5-33
deleting 5-36
recording states 5-34
states 5-33
viewing in data recorder 5-34
Script Manager 7-30
playback 7-31
recording 7-30
Scroll Bars 1-26
Selecting Items 1-19
Show Name Only 4-10
Simulation Basis Environment
See Environments.
Simulation Menu
enter basis environment 7-15
event scheduler 7-14
integrator 7-14
main properties 7-11
Oil Output Settings 7-16
optimizer 7-13
view equations 7-15
Snap Shot
See Case.
See Printing.
Sorting Information 4-20
Sounds 7-56
Specsheets
printing 6-5
workbook 6-5
Starting a Simulation 1-30
using the menu bar 7-4
States
See Case.
Stream Configuration Preferences 7-57
Streams
adding utilities 5-49
allowing multiple connections 7-34
analysis 5-4
bend points 3-48
copy conditions 1-38
deleting 1-51
deleting from workbook 4-10
installation example 1-38
installing 1-38, 7-17
installing from object palette 1-34
installing from workbook 4-5
manual routing 3-47
package properties 7-57
PFD label options 3-21
property view 1-9
quick route mode 3-46
rerouting on PFD 3-46
spec stream as 1-37
view new upon creation 7-34
Strip Charts
adding 5-21

curves 5-24
deleting 5-22
graph control 5-27
graphical line properties 5-30, 5-31
historical data 5-32
interval markers 5-25
manipulating 5-24
object inspection 5-26
viewing 5-22
zooming 5-26
Sub-Flowsheets
viewing from workbook 4-9
Subflowsheets
advantages 2-11
capabilities 2-11
components 2-11
connections 2-15
installing 2-22
parameters 2-17
property view 2-15
transfer basis 2-18
variables 2-21
Summary View 1-8, 1-52
Swapping Connections in the PFD 3-7

T

Template Tag 2-33
Templates 2-32
accessing PFDs 3-23
creating 2-36
exported connections 2-33
exported variables 2-35
feed and product streams 2-34
information 2-32
installed simulation basis 2-33
installing 2-37
reading 2-38
reading existing 2-22
refrigeration loop example 2-39
Terminology
Structure 1-10
Tool Commands 7-27
Tool Tips Preferences 7-37
Tools Menu
DCS 7-29
face plates 7-28
macro language editor 7-32
preferences 7-32
script manager 7-30
Trace Window 5-46
object inspection 5-48
Transfer Basis
flash types 2-34
See Flowsheet.
Tray Sizing
% Liquid Draw 8-91
Use Tray Vapour to Size 8-91
Tray Sizing Preferences 7-57
Trayed Sections
See Column Sizing Utilility.
True and Pseudo Critical Properties
See Critical Properties Utility.

U

Unit Sets
adding conversions 7-42
adding new 7-40
deleting conversions 7-43
viewing conversions 7-42
Units Preferences 7-40
User Properties 7-21
adding 7-24
User Properties Utilility 8-109
User Properties Utility
generalized procedure 8-111
quick start 8-109
Utilities
adjust operation target 8-118
attaching 5-48
available utilities 8-3
boiling point curves 8-4
procedure 8-4
cold properties 8-12
column sizing 8-79
critical property 8-14
deletion 5-53
depressuring 8-17
envelope 8-40
hydrate formation 8-45
parametric 8-52
pinch 8-65
pipe sizing 8-69
property table 8-73
tray sizing 8-79
user properties 8-109
vessel sizing 8-114
viewing 5-52

V

Valve Equations
See Depressuring Utility.
Variable Navigator 5-11
navigator scope 5-13
Vessel Sizing Utility
cost analysis 8-116
generalized procedure 8-115
quick start 8-114
View Equations 7-15

W

Wetted Area
See Depressuring Utility.
Window
menu bar commands 7-60
Windows Functionality 1-4
Workbook
accessing objects 4-6
deleting objects 4-10
installing a stream 4-5
installing operations 4-5
new tabs 4-12
object sort 4-20
opening 4-4
page setup 4-11
re-order variables 4-17
selecting variables 4-15
tables on PFD 3-26
variable sets 4-17
variable sort 4-21
view 1-7
Workbook Menu
commands 7-25
Workbook Tabs
deletion 4-18
editing 4-14
exporting/importing 4-18
new 4-12
Workspace
loading 7-61

Z

Zoom Buttons
PFD 3-5
preview 6-6