

UniSim® Design

User Guide

Honeywell

Copyright

April R390 Release

The information in this help file is subject to change over time. Honeywell may make changes to the requirements described. Future revisions will incorporate changes, including corrections of typographical errors and technical inaccuracies.

For further information please contact

Honeywell
300-250 York Street
London, Ontario
N6A 6K2
Telephone: (519) 679-6570
Facsimile: (519) 679-3977

Copyright Honeywell 2009. All rights reserved.

Prepared in Canada.

Table of Contents

1	Interface	1-1
1.1	Introduction	1-2
1.2	Interface Basics	1-3
1.3	Object Status Window/Trace Window	1-11
1.4	Toolbar	1-14
1.5	Hot Keys.....	1-16
2	Program Philosophy	2-1
2.1	Introduction	2-2
2.2	Simulation Case	2-5
2.3	Multi-Flowsheet Architecture/Environments.....	2-7
3	Flowsheet.....	3-1
3.1	Introduction	3-2
3.2	Flowsheets in UniSim Design	3-2
3.3	UniSim Design Environments	3-4
3.4	Sub-Flowsheet Environment	3-10
3.5	Templates	3-14
3.6	Property View Flowsheet Analysis	3-22
4	File Management	4-1
4.1	Menu Bar	4-2
4.2	File.....	4-2
4.3	UFL Files	4-11
5	Basis Environment.....	5-1
5.1	Introduction	5-2
5.2	Simulation Basis Manager.....	5-2
5.3	Reaction Package	5-25
5.4	Component Property View	5-26
6	Oil Characterization Environment	6-1
6.1	Introduction	6-2
6.2	Oil Characterization Manager	6-2
7	Simulation Environment	7-1
7.1	Introduction	7-5

7.2	Main Properties	7-5
7.3	UniSim Design XML	7-10
7.4	Optimizer	7-12
7.5	Event Scheduler	7-13
7.6	Integrator	7-26
7.7	Adjust-Recycle Manager	7-28
7.8	Initialize From	7-37
7.9	Dynamic/Steady State Modes	7-50
7.10	Solver Active/Holding.....	7-51
7.11	Integrator Active/Holding	7-51
7.12	Equation Summary.....	7-52
7.13	Enter Basis Environment	7-52
7.14	User Variables	7-52
7.15	Importing & Exporting User Variables.....	7-54
7.16	Oil Output Settings.....	7-55
7.17	Object Navigator	7-56
7.18	Simulation Navigator	7-58
7.19	Notes Manager.....	7-60
7.20	Optimization Objects	7-61
7.21	Reaction Package	7-63
7.22	Fluid Package/Dynamics Model.....	7-63
7.23	Workbook	7-64
7.24	PFD	7-77
7.25	Column.....	7-94
7.26	Utilities	7-94
7.27	Simulation Balance Tool	7-98
8	UniSim Design Objects	8-1
8.1	Installing Objects	8-3
8.2	Defining Objects	8-5
9	Print Options	9-1
9.1	Introduction	9-2
9.2	Printing in UniSim Design	9-2
9.3	Reports.....	9-7
9.4	Printing the PFD as a File.....	9-14
10	Edit Options.....	10-1
10.1	Introduction	10-3
10.2	Edit Menu.....	10-3
10.3	Editing the PFD	10-4
10.4	Graph Control.....	10-36

10.5 Format Editor	10-41
11 Simulation Tools.....	11-1
11.1 Introduction	11-3
11.2 Workbook	11-3
11.3 PFD	11-3
11.4 Case Summary	11-3
11.5 Utilities	11-4
11.6 Reports.....	11-4
11.7 Databook	11-4
11.8 Face Plates.....	11-39
11.9 Dynamics Assistant	11-40
11.10Control Manager	11-41
11.11Dynamic Profiling Tool	11-41
11.12Snapshot Manager	11-44
11.13Script Manager	11-50
11.14Macro Language Editor.....	11-52
11.15Case Security	11-54
11.16Echo ID	11-70
11.17Correlation Manager	11-71
11.18Alarm Manager	11-85
11.19Variable Navigator.....	11-88
11.20Simulation Balance Tool	11-90
11.21RTO Manager.....	11-90
12 Session Preferences	12-1
12.1 Introduction	12-3
12.2 Simulation Tab.....	12-5
12.3 Variables Tab.....	12-20
12.4 Reports Tab.....	12-26
12.5 Files Tab	12-30
12.6 Resources Tab	12-36
12.7 Extensions Tab	12-42
12.8 Oil Input Tab	12-43
12.9 Tray Sizing Tab.....	12-44
12.10Case Tools Tab	12-48
13 Window & Help Options.....	13-1
13.1 Introduction	13-2
13.2 Window Menu	13-2
13.3 Help Menu.....	13-4

Index.....	I-1
-------------------	------------

1 Interface

1.1 Introduction	2
1.1.1 Event Driven	2
1.1.2 Modular Operations	2
1.1.3 Multi-flowsheet Architecture	2
1.1.4 Object Oriented Design.....	3
1.2 Interface Basics.....	3
1.2.1 Views Functionality.....	3
1.2.2 Primary Interface Elements	3
1.2.3 Multi-Flowsheet Architecture/Environments	4
1.2.4 Navigators	5
1.2.5 Objects.....	6
1.2.6 Structure Terminology	6
1.2.7 Desktop.....	7
1.2.8 Interface Terminology	9
1.3 Object Status Window/Trace Window	11
1.3.1 Opening & Sizing the Windows.....	11
1.3.2 Message Windows	12
1.3.3 Object Inspect Menu	12
1.4 Toolbar	14
1.4.1 Dynamics Time Manager Tool Bar Icons	16
1.5 Hot Keys	16

1.1 Introduction

UniSim Design offers a high degree of flexibility because there are multiple ways to accomplish specific tasks. This flexibility combined with a consistent and logical approach to how these capabilities are delivered makes UniSim Design an extremely versatile process simulation tool.

The usability of UniSim Design is attributed to the following four key aspects of its design:

- Event Driven operation
- Modular Operations
- Multi-flowsheet Architecture
- Object Oriented Design

1.1.1 Event Driven

This concept combines the power of interactive simulation with instantaneous access to information. Interactive simulation means the information is processed as it is supplied and calculations are performed automatically. Also, you are not restricted to the program location where the information is supplied.

1.1.2 Modular Operations

Modular Operations are combined with the Non-Sequential solution algorithm. Not only is information processed as it is supplied, but the results of any calculation are automatically produced throughout the flowsheet, both forwards and backwards. The modular structure of the operations means they can be calculated in either direction, using information in an outlet stream to calculate inlet conditions. Process understanding is gained at every step because the operations calculate automatically and results are seen immediately.

1.1.3 Multi-flowsheet Architecture

Multi-flowsheet architecture can be used to create any number of flowsheets within a simulation and to easily associate a fluid package with a defined group of unit operations.

1.1.4 Object Oriented Design

The separation of interface elements (how the information appears) from the underlying engineering code means the same information appears simultaneously in a variety of locations. Each display is tied to the same process variable, so if the information changes, it automatically updates in every location.

Also, if a variable is specified, then it is shown as a specification in every location. This means the specification can be changed wherever it appears and you are not restricted to a single location for making changes.

1.2 Interface Basics

This section provides basic information about using the UniSim Design interface.

1.2.1 Views Functionality

UniSim Design has the same basic features as found in other Windows 2000 or XP based programs:



Minimize icon



Maximize icon



Restore icon



Close icon



Pin icon

- Minimize, Maximize/Restore and Close icons are located in the upper right corner of most views.
- Object icon, located in the upper left corner of most views, contains the normal Windows 3.x menu.

Most of the different views found in UniSim Design are resizable to some degree.

The following list provides a brief description on resizable views:

- When the Minimize, Maximize/Restore and Close icons are available, the view can be resized vertically and horizontally.
- When only the Minimize and Close icons are available, the view can only be resized vertically.
- When only the Close icon or Close and Pin icons are available, the view can not be resized.

1.2.2 Primary Interface Elements

Although you can input and access information in a variety of ways, there are five primary interface elements for interacting with UniSim

Design:

Interface Element	Description
PFD	A view containing a graphical environment for building your flowsheet and examining process connectivity. Process information can be displayed for each individual stream or operation as needed.
Workbook	A view containing a collection of tabs that displays information in a tabular format. Each Workbook tab displays information about a specific object type. You can install multiple tabs for a given object type, displaying information in varying levels of detail.
Property View	A single view that contains multiple tabs. UniSim Design extensively uses these single views, which include all information about a specific object (i.e., an individual stream or operation).
Summary View	Displays the currently installed streams and operations.
Simulation Navigation	A view that provides a single location for viewing all stream and unit operation property views in the simulation case, regardless of the flowsheet they exist in.

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. Changes made in any location are automatically reflected throughout UniSim Design.

In addition, there are no restrictions as to 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.

1.2.3 Multi-Flowsheet Architecture/Environments

UniSim Design is developed around a Multi-flowsheet Architecture. After creating the fluid package(s) for the simulation, you enter the main flowsheet. This is where the bulk of the model is created (where you install the streams and operations that represent the process).

Sub-flowsheets can also be created at any time within the main flowsheet. Sub-flowsheets appear as a single operation with multiple connections. The main simulation does not know what is inside the sub-flowsheet, meaning it could be a refrigeration loop or a decanter system. The sub-flowsheet is seen by UniSim Design as any other operation and it calculates whenever conditions are changed within it.

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 enter the sub-flowsheet's Build environment, the following occurs:

- The main flowsheet is temporarily cached and hidden; it returns to the exact status when you exit the sub-flowsheet.
- Other flowsheet solvers still produce the effect of a change, but the results of that change are not produced beyond the flowsheet boundary until you leave the sub-flowsheet environment.

Example: A stream inside of a sub-flowsheet containing a flow specification can also be connected to a stream in the parent flowsheet. If changes are made to the flow rate in the sub-flowsheet environment, the flow in the parent flowsheet is forgotten, as are any other flows in any flowsheets that are calculated as a result of that flow specification. This is the "forget" pass in the UniSim Design solver.

Considering the "forget" pass in the UniSim Design solver, the definition of a flowsheet (or sub-flowsheet) in the context of the overall program is defined by what it possesses:

- Independent fluid package (optional)
- PFD
- Workbook
- Flowsheet Elements (streams and/or operations)
- Solver

This definition may seem to contradict previous statements regarding the access to information, however, capabilities were built into UniSim Design 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 that allows you to access information inside the sub-flowsheet without ever entering the sub-flowsheet itself.

1.2.4 Navigators

All of the flowsheets within a simulation are tied together through the

Refer to [Section 7.18 - Simulation Navigator](#) for details on the Simulation Navigator.

Refer to [Section 7.17 - Object Navigator](#) for details on the Object Navigator.

Refer to [Section 11.19 - Variable Navigator](#) for details on the Variable Navigator.

Navigators.

Navigators	Description
Simulation Navigator	Quick access to the property view of any stream or unit operation from any flowsheet within the case.
Object Navigator	Immediate access to the property view for any stream or operation from any location. You can access the Object Navigator view by right-clicking on any blank area of the UniSim Design Desktop and selecting Find Object command from the Object Inspect menu.
Variable Navigator	Target process variables from any flowsheet. For example, you can select variables for inclusion on a Strip Chart or for attachment to logical operations such as Adjusts or Controllers.

1.2.5 Objects

The term object is used extensively throughout the documentation to refer to an individual stream or operation. Within UniSim Design, information associated with an object can appear in a variety of ways (Workbook, PFD, Property View, Plot, etc.).

Through the object oriented design of UniSim Design, 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.

Objects, such as an icon in a PFD, are tied to appropriate commands for that object (i.e., printing, direct access to a property view, etc.).

1.2.6 Structure Terminology

All UniSim Design cases include certain structural elements. The following table defines some common UniSim Design terminology.

Object	Definition
Flowsheet Element (or Object)	A Stream or Operation.
Flowsheet	A collection of Flowsheet Elements that utilize a common fluid package. 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.

Object	Definition
Simulation Case	A collection of fluid package(s), flowsheets, and flowsheet elements that form the model. The simulation case can be saved to disk for future reference. The extension used for saved cases is *.usc. Simulation cases can also be saved as template files (*.tpl), UFL files (*.ufl) and XML files (*.xml).
Session	Encompasses every simulation case that is open while UniSim Design is running.

Special Flowsheet Elements

Column operations and flowsheet templates are special Flowsheet Elements because 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 as follows:

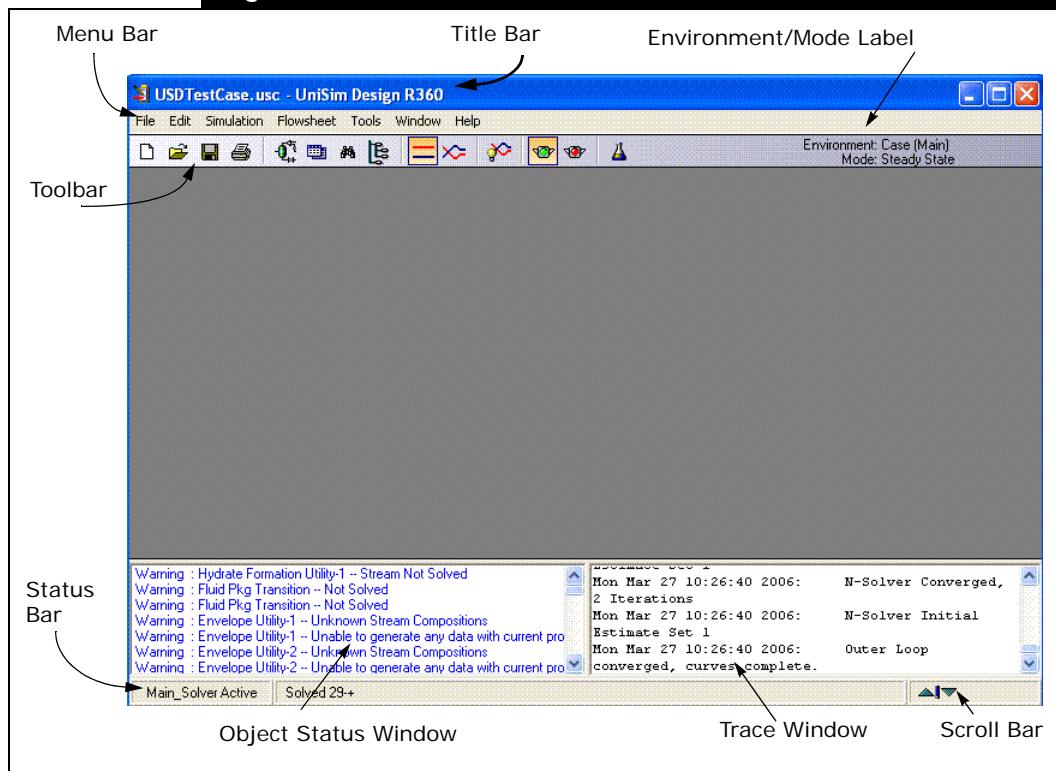
- Contain their own flowsheet, meaning they possess their own PFD and Workbook.
- Can be comprised of multiple flowsheet elements.
- Can be retrieved as a complete entity into any other simulation case.

1.2.7 Desktop

The figure below shows the basic components of the UniSim Design

Desktop.

Figure 1.1



The main features of the Desktop are defined in the following table.

Object	Definition
Title Bar	Indicates the UniSim Design file currently loaded.
Menu Bar	Provides access to common flowsheet commands through a drop-down menu system.
Toolbar	Contains various icons that invoke a specific command when clicked.
Environment/Mode Label	Indicates the environment and mode that you are currently working in.
Status Bar	Displays the calculation status of the object. When the mouse pointer is placed over an icon in the toolbar, the Object Palette, or a property view, a brief description of its function appears in the Status Bar.
Calculation/Responsiveness icon	The Calculation/Responsiveness icon enables the user to control how much time is spent updating the screens vs. calculations.



Calculation/
Responsiveness icon

Object	Definition
Scroll Bars	Allows you to scroll horizontally and vertically.
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. For more information about the Object Status Window or Trace Window, refer to Section 1.3 - Object Status Window/Trace Window .

Some additional information about the UniSim Design Desktop:

- When the mouse pointer is placed over a button, its descriptive name pops up below the pointer and a Fly by function appears in the status bar.
- When necessary, the Desktop has both a vertical and horizontal scroll bar that are automatically created.

1.2.8 Interface Terminology

The terminology shown in the following figures is used to describe the various UniSim Design interface elements.

Figure 1.2

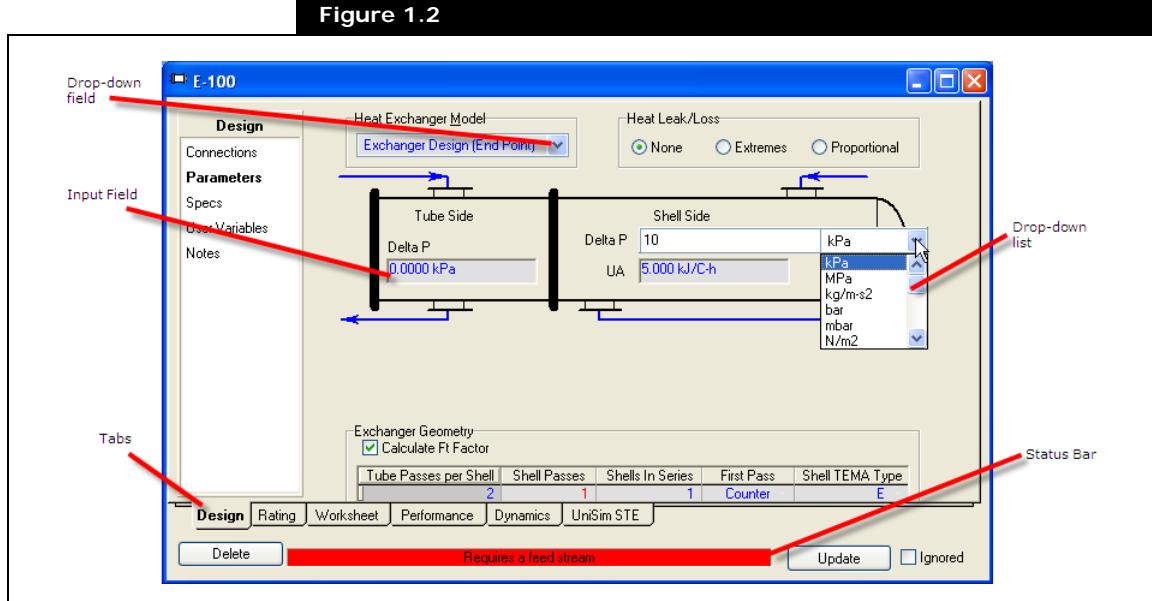
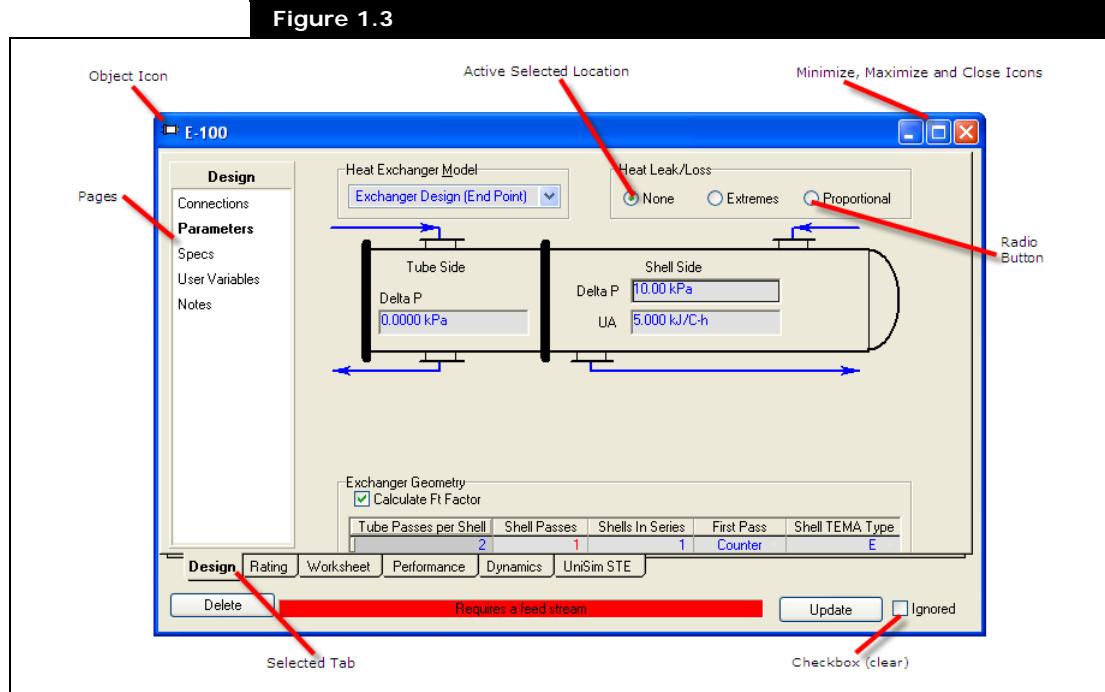


Figure 1.3



Object	Definition
Active Selected Location	The current active location is always indicated by a dark frame or border.
Button	Invokes a command when clicked.
Checkbox	Items or settings that are On or Off. Checking the checkbox turns the function On. Unchecking it turns it Off.
Drop-Down List	A list of available options for a given input cell.
Group	Organizational border within a page that groups related functions together. Each group has its own active location.
Icon	Invokes a command when clicked, or opens a view when double-clicked.
Input Cell/Field	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.
Matrix	A group of cells where you can manoeuvre with the mouse or the keyboard arrow keys.
Minimize/Maximize icon	Either shrink the current view (minimize), or expand the view to its full size.
Object icon	Either closes the view (double-clicking), or produces a drop-down menu of common Windows commands.
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).
Pages	Provides access to detailed information for the selected object.
Pin	Converts a Modal property view to a Non-Modal property view.

Object	Definition
Radio Button	Always found in groups of at least two; only one can be active at a time.
Tabs	Provides a logical grouping of information. Often contain pages where the information is sorted further.
View	Any graphical representation found on the Desktop, for example, a property view for an operation.

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 selected. Within that view, there is again only one location that is Active. How this appears varies depending on the location (cell, button, etc.).

1.3 Object Status Window/ Trace Window

At the bottom of the UniSim Design Desktop there is a window that appears by default. The window is split vertically into two panes and 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.

The Object Status Window and Trace Window cannot be opened separately.

1.3.1 Opening & Sizing the Windows

To open the Object Status and Trace Windows, position the mouse pointer on any part of the 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 that 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.

1.3.2 Message Windows

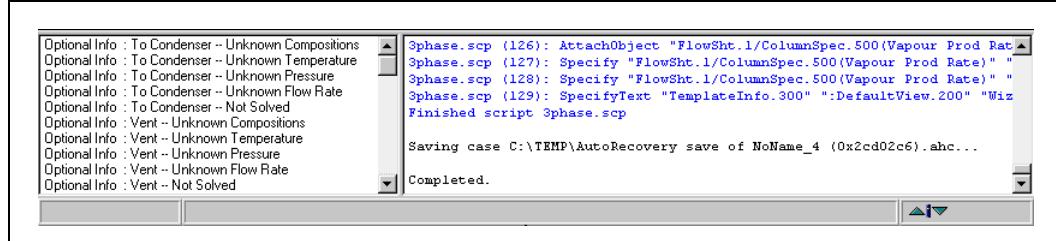
Status messages displayed in yellow in a property view appear in black in the Object Status Window for clarity.

The message windows within UniSim Design include the Object Status and Trace windows. Refer to the following table for the functionality of the windows:

Window	Functionality
Object Status Window	<ul style="list-style-type: none"> Shows current status messages for objects in the flowsheet, coloured accordingly. The colour of the status message for an object usually matches the colour of the status message on the object's property view. Allows you to access to the property view of an object described in the status message by double-clicking on the message.
Trace Window	<ul style="list-style-type: none"> Displays iterative calculations for certain operations (such as the Adjust, Recycle, Reactor, etc.). These appear in black text. Displays scripting commands in blue text. Displays error messages (that still solve), such as operation errors or warnings, in red text.

An example of the contents shown in the Object Status and Trace Windows appears below. Each window has a vertical scroll bar for viewing the contents of the window.

Figure 1.4



1.3.3 Object Inspect Menu

The commands available through the Object Inspect menu of the Object Status Window and Trace Window are specific to each pane.

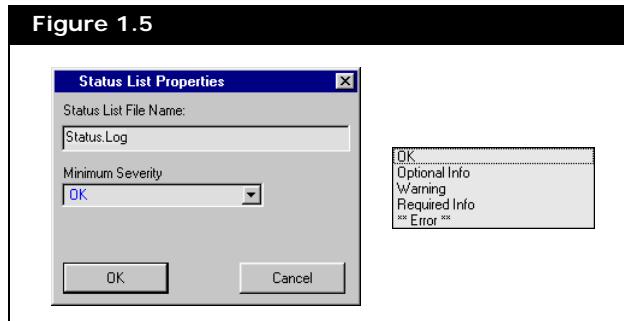
Object Status Window

Status messages that are **OK** do not appear in the Object Status Window.

The following commands are available by right-clicking the Object

Status Window:

Command	Description
View Status List Properties	Opens the Status List Properties view. This view contains an input field for the Status List File Name (by default Status.Log), that enables 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 example, selecting Warning from the list displays all messages that are warnings or more severe in the left pane. To display only error messages that are the most severe, select the **Error** option.
Dump Current Status List to File	Automatically dumps the contents of the left pane to the Status List File Name.

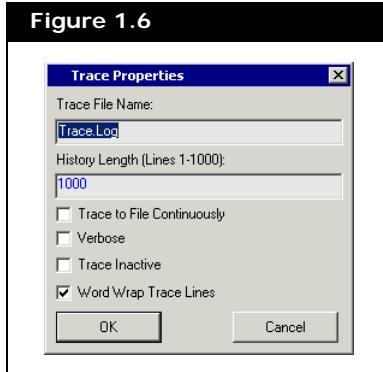


Trace Window

The commands in Object Inspect menu for the Trace Window are described in the following table:

Command	Description
View Trace Properties	Opens the Trace Properties view, which contains the following: <ul style="list-style-type: none"> Trace File Name field. Shows the file name to which the contents of the Trace Window can be written (by default Trace.Log). History Length field. Represents the number of lines that the Trace Window keeps in its history. Trace to File Continuously checkbox. When checked, the Trace Window contents are written to the Trace File. Verbose checkbox. When checked, the Trace Window shows solver information for all operations in the case. Trace Inactive checkbox. When checked, the Trace Window shows information for all inactive operations in the case. Word Wrap Trace Lines checkbox. When selected, the messages in the Trace Window are word wrapped to fit the Trace Window.
Set trace window font	Change the trace window default font

Command	Description
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.
Copy all trace information	Copy all trace information to the clipboard



1.4 Toolbar

These commands are also available in the menu bar.

The icons on the toolbar provide immediate access to the most commonly used commands.

The toolbar varies depending on the current environment and Mode.

The following buttons are found on the various tool bars in UniSim Design.

Name	Icons	Description
New Case		Creates a new case.
Open Case		Locates and opens an existing case/template/column.
Save Case		Saves the active case.
PFD		Opens the PFD for the current flowsheet.
Workbook		Opens the Workbook for the current flowsheet.
Navigator		Opens the Object Navigator.
Simulation Navigator		Opens the Simulation Navigator.

Name	Icons	Description
Steady State/ Dynamics		Toggles between Steady State and Dynamic modes. Currently toggled to Steady State mode.
Dynamics Assistant		Opens the Dynamics Assistant view.
Column		Opens the Column Runner view.
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. Red (right) is Holding.
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 about the UniSim Design Desktop:

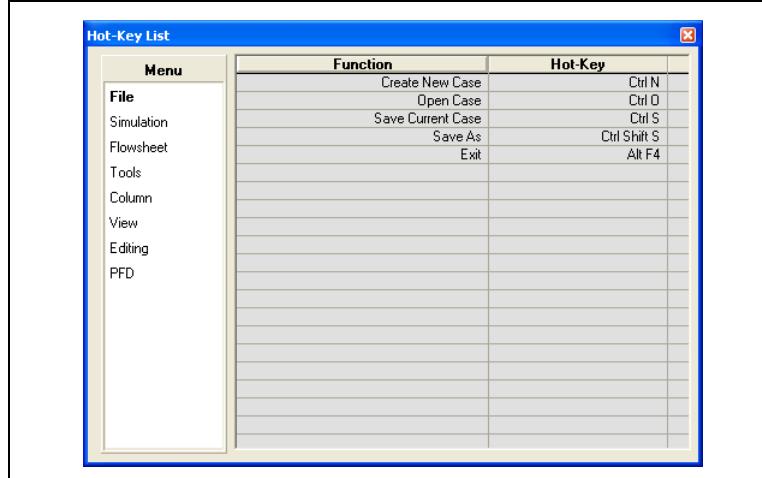
- When the cursor is placed over a button, its descriptive name pops up below the pointer and a Fly by function appears in the status bar.
- The Desktop has both a vertical and horizontal scroll bar. These are automatically created when necessary.

1.4.1 Dynamics Time Manager Tool Bar Icons

Name	Icons	Description
Take One Step		This causes the simulation to take one time step. If the red light is currently on, pressing this button will first turn the green light on. If the Integrator is currently in Automatic, pressing this button will change to Manual mode.
Real Time		This button activates the real time checkbox on the Integrator.
Desired Real Time Factor		This provides a pop-up window to enter the Desired Real Time Factor on the Integrator.
Take Multiple Steps		When pressed the integrator switches to run and manual mode, if required, and executes the requested number of steps. See the Set Multiple Steps icon below.
Set Multiple Steps		Allows the user to enter the number of steps to execute. See Take Multiple Steps above.
Integrator In Auto		Switches the integrator between Auto and Manual mode.
Alarm Manager		Opens Alarm Manager. Used to view and acknowledge current status of raised alarm events by suitably-configured equipment items.

1.5 Hot Keys

Figure 1.7



The Hot Key list is available under the Help menu of the main screen.

File	
Create New Case	CTRL N
Open Case	CTRL O
Save Current Case	CTRL S
Save As	CTRL SHIFT S
Exit UniSim Design	ALT F4
Simulation	
Enter Simulation Basis Manager	CTRL B
Main Properties	CTRL M
Access Optimizer	F5
Access Event Scheduler	CTRL E
Leave Current environment (Return to Previous)	CTRL L
Toggle Steady-State/Dynamic Modes	F7
Toggle Hold/Go Calculations	F8
Access Integrator	CTRL I
Start/Stop Integrator	F9
Take one step integration	CTRL 1
Stop Calculations	CTRL BREAK
Flowsheet	
Add Material Stream	F11
Add Operation	F12
Access Object Navigator	F3
Access Notes Manager	CTRL G
Show/Hide Object Palette	F4
Access Composition View (from Workbook)	CTRL K
Displays Stream Temperatures	SHIFT T
Displays Stream Pressures	SHIFT P
Displays Stream Molar Flow Rates	SHIFT F
Displays Stream Names (Default)	SHIFT N
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 Dynamics Assistant	CTRL Y
Access Help	F1
Column	
Go to Column Runner (sub-flowsheet)	CTRL T
Stop Column Solver	CTRL BREAK
View	

Close Active View	CTRL F4
Tile Views	SHIFT F4
Go to Next View	CTRL F6 or CTRL TAB
Go to Previous View	CTRL SHIFT F6 or CTRL SHIFT TAB
Go to Next item within View	TAB
Go to Previous item within View	SHIFT TAB
Editing/General	
Access editing cell function	F2
Access Pull-Down Menus	F10 or ALT
Go to Next Page Tab	CTRL SHIFT N
Go to Previous Page Tab	CTRL SHIFT P
Undo	CTRL Z
Cut	CTRL X
Copy	CTRL C
Paste	CTRL V
PFD	
Zoom out	Page Down
Zoon in	Page Up
Zoom All	Home
Toggle between last zoon levels	Z
Pan	Arrow or SHIFT Arrow Keys
Centre PFD on cursor	Period key or C
Mirror about X axis	X
Mirror about Y axis	Y
Rotate 90	1
Rotate 180	2
Rotate 270	3
To default orientation	N
Display Stream temperatures	SHIFT T
Display Stream pressures	SHIFT P
Display Stream molar flowrates	SHIFT F
Display Stream names	SHIFT N
Dsplay Stream mass flowrates	SHIFT M
Display Stream descriptions	SHIFT R
Display Outlet nozzle elevation	SHIFT O
Display Inlet nozzle elevation	SHIFT I
Select Object label	L
Open Selected Object	V or E
Delete Selected Object	Delete

2 Program Philosophy

2.1 Introduction	2
2.1.1 One Model, Many Uses.....	2
2.1.2 The Leader in Usability	3
2.1.3 Maximizing the Engineer's Efficiency	4
2.1.4 The Difference in the UniSim Design Series	5
2.2 Simulation Case	5
2.2.1 Building a Simulation Case	6
2.3 Multi-Flowsheet Architecture/Environments	7

2.1 Introduction

UniSim Design is based on these fundamental principles:

- “single model” concept
- rigorous first principle’s models
- reuse of simulation data
- best in class usability
- an open customizable environment

The UniSim Design series of products is true to these principles as new developments have expanded and built upon this foundation.

2.1.1 One Model, Many Uses

The single model concept allows the user to build one model of the process and migrate it through the various stages of the Lifecycle. During the design stage, a model can be used for conceptual design, real process design, detailed engineering design, and finally, for operability design. Once the asset has been built, this same model can be used for operations improvement, operator training, safety studies, and asset optimization. In addition to delivering simulation capabilities that support the Lifecycle, UniSim Design also serves as the platform for modeling across the entire range of the chemical and hydrocarbon processing industries.

UniSim Design serves as the engineering platform for modeling processes from Upstream, through Gas Processing and Cryogenic facilities, to Refining and Chemicals processes. A range of powerful new engineering capabilities have been delivered within UniSim Design through the development activities of Honeywell and its alliance partners, including the following:

- **UniSim Thermo.** Completely flexible thermodynamics, UniSim Thermo, is the first truly “componentized” thermodynamics server now on the market in the process industries. UniSim Thermo is a thermodynamic calculation framework that makes it possible to develop independent, extensible, and encapsulated calculation modules for reuse within the engineering Lifecycle. The immediate benefits are more flexible thermodynamic choices and the easy integration of in-house and 3rd party methods.
- **UniSim Design OLI Interface.** Integration of OLI Systems Inc. technology and component databanks with the industry’s first fit for purpose set of electrolyte unit operations.
- **Multi-phase Pipeline Hydraulics.** Integration of PIPESYS from Neotechnology.
- **Transient Multi-phase Flow.** Integration with Honeywell’s ProFES tools.

- **Advanced Amine Systems.** Integration of AMSIM from DB Robinson.

Beyond the development and integration of these technologies into the UniSim Design engineering platform, there have been many new capabilities developed that enhance Process Asset Lifecycle Management. These include the following:

- **Honeywell SQP.** A new optimization algorithm, Honeywell SQP, for design and asset optimizations.
- **LP Utility.** A utility that uses the rigorous UniSim Design process model to generate vector data for planning and scheduling tools.
- **Dynamic Depressuring.** Uses the powerful, proven dynamic modelling capabilities of UniSim Design for conducting depressuring studies in our steady state simulator to perform Relief Valve sizing against safe depressuring times.
- **LNG Rating.** Integration of the rigorous UniSim PFE engine from the UniSim Heat Exchanger software.
- **Air Cooler Rating.** Integration of the UniSim CFE functionality from the UniSim Heat Exchanger software.
- **Heat Exchanger Rating.** Tight integration of the UniSim STE engine from the UniSim Heat Exchanger software so that process engineers can perform detailed rating calculations within the UniSim Design environment.
- **SPS.** Integration of solid component characterization technology from SPS.

2.1.2 The Leader in Usability

Honeywell has always believed that ease of use is a fundamental component of simulation technology. UniSim Design has consistently delivered on this, allowing engineers to easily construct and analyze models of their process to obtain the understanding necessary to make informed engineering, operating and business decisions.

As with previous releases of UniSim Design, there are a range of new features that simplify the engineer's task of building and analyzing models. Some of the highlights include:

- A Simulation Navigator that allows instant access to all unit operations and streams within the simulation case.
- Case Collaboration that allows the building of compound cases that span the user's network. Boundary streams are connected through an external data server which supports revisions and change notifications.
- A Notes Manager that allows one spot access to all user notes from any location in the simulation case.
- Auto Connection feature for rapidly building flowsheets in the PFD environment.
- Correlation Sets, which allow the user to define the properties to be calculated and displayed for any stream in the case.

- Case security levels that allow the protection of Intellectual Property contained within any built UniSim Design model.

2.1.3 Maximizing the Engineer's Efficiency

There are several key aspects of UniSim Design which have been designed specifically to maximize the engineer's efficiency in using simulation technology. Usability and reliability are two obvious attributes, which UniSim Design has and continues to excel at. The single model concept is key not only to the individual engineer's efficiency, but to the efficiency of an organization.

Of equal importance is the commitment to developing the capabilities within the simulator that support re-use of the engineer's work, as well as capabilities that allow for the flexible application of the available technology. The UniSim Design 350 series contains a number of new developments which are designed specifically to promote re-use and deliver increased flexibility to the engineer. Underlying this aspect of UniSim Design has been a steady migration to delivering finer granularity of the "components" which make up a simulation case, such that these components can be defined once and used many times.

The most significant development within the UniSim Design 350 series in this regard is the delivery of XML (eXtensible Mark up Language) technology. The range of possibilities that this opens up are significant, but some of the immediate benefits are:

- The ability to store all (or part) of the user's inputs and specifications in XML to allow re-building of the case.
- The ability to store parts of an existing simulation case in XML and have it read into another case, either augmenting or overwriting the definitions within that case.
- The ability to store simulation case results in an XML format to allow post processing of simulator data, taking advantage of the wide range of XML technology being developed within the software industry.
- The ability to browse the simulation case data in a familiar internet browser-like environment.

In addition, UniSim Design has increased ability to define and store simulation components, including:

- **Workbook Sheet definitions.** Individual pages of the Workbook can be stored out and read back into any other simulation case.
- **Correlation Sets.** User defined sets of properties can be configured and read into any other case.
- Independent and Dependant Property Sets for LP utilities.

- **UFL files.** Unit operation collections from an existing case can be stored out and modified as *.ufl files, allowing them to be re-imported via the copy/paste capabilities into any other case.

2.1.4 The Difference in the UniSim Design Series

For existing UniSim Design users there are some significant differences in the UniSim Design series that the user will want to take advantage of. There have been significant advancements in the underlying “fluid” structure within UniSim Design that has delivered a range of exciting possibilities.

- A “component” (library or pseudo) has been defined. The component within UniSim Design has become both flexible and extensible. Properties of components can change through the flowsheet, either through user intervention or via the action of a unit operation.
- Component lists can be shared amongst fluid packages, which, in combination with the flexible component technology, significantly reduces the number of components required to model a given process, particularly in refining.
- User properties have been integrated into the components and have been tied into flexible stream reporting capabilities.
- Multiple fluid packages are supported within a single flowsheet. The user is not restricted to having one fluid package per flowsheet. Fluid packages can be applied to individual streams and operations within a flowsheet, with Fluid Package Transitions automatically inserted (or removed) where necessary.

New Technology Built on Historical Success

The UniSim Design series represents a significant advancement in simulation technology, built upon the proven capabilities of previous versions. As with every Honeywell product, it reflects our commitment to Plant Asset Lifecycle Management by delivering the best tools within a platform that is the world leader in ease of use and flexibility, and sets the standard for an open engineering environment.

2.2 Simulation Case

The simulation case is comprised of the main elements described in the

following table:

Main Elements	Description
Fluid Definition	The definition of the material that is being operated on by the unit operations, including component lists, component properties, property package, reactors, etc. Refer to Chapter 5 - Basis Environment for more information.
Flowsheet	A collection of unit operations (physical and logical) and the streams that connect fluid information between them. This is termed Topology and connectivity. See Chapter 3 - Flowsheet for more information.
Analytical Calculations	The property calculations (stream based) and utilities that perform additional calculations using data (typically stream information) owned by other objects. Refer to Appendix A - Property Methods & Calculations in the UniSim Design Simulation Basis Guide for more information.
Data Sources	Variables which are owned by the unit operations can either be used by other unit operations (logical operations) in their calculations, or attached to Data Collectors for visualization. Refer to Section 11.19 - Variable Navigator for more information.
Data Collectors	Elements within the program that access data owned by other objects for the purpose of visualization or analysis. Refer to Section 11.7 - Databook for more information.
Simulation Control Tools	These are the tools that sit on top of the simulation case, causing it to solve in a specific manner to deliver specific behaviour. Refer to the following objects Optimizer, Dervutil, Data Recon utility, PM utility, and Case Studies in the UniSim Design Operations Guide , and Section 7.6 - Integrator .

2.2.1 Building a Simulation Case

If you use the basic steps of building a simulation case, the ability to reuse these simulation elements can be more easily illustrated:

1. Create the fluid definition. Fluid packages can be stored as self-contained pieces (as well as some of the pieces within the fluid package) and read in to begin a simulation case.
 2. Construct the flowsheet topology. Flowsheet templates, *.ufl files, and *.xml files can all be stored as self-contained pieces and be read in to any future simulation case. With UniSim Design, changes made in the external files (*.ufl and *.xml) can be easily incorporated into an existing case.
- Then optionally:
3. Define the Property Calculations wanted for the various fluid types. The correlation sets or Workbooks (the entire Workbook definition or individual tabs/pages) can be stored outside of the simulation case and subsequently applied to existing or new cases.

4. Create any Analytical Calculators required.
5. Identify any data sources that are required for Data Collectors.
6. Define any Data Collectors (Strip charts, Workbooks, PFD tables).
7. Construct any Simulation Control Tools required (Optimizer, Integrator, or Case Study).

2.3 Multi-Flowsheet Architecture/ Environments

With the continued evolution of computer hardware and software architecture, the ability to rigorously model entire plants has become feasible. UniSim Design, which has always been based on a multi-flowsheet architecture, is ideally suited for dealing with the size of the simulation cases that result from building plant-wide models.

Once the fluid package(s) for your simulation have been created, you enter the main flowsheet. In this location, the bulk of the model is created, installing the streams and operations to represent your process.

There are two types of sub-flowsheets: columns and templates.

Sub-flowsheets within the main flowsheet can be created at any time, as well as sub-flowsheets within sub-flowsheets. There are three fundamental purposes of the sub-flowsheet:

- Representation of complex plant models in terms of “units” which provides an easy mechanism for the organization of large models.
- Easy support for templating of units or processes to facilitate their re-use.
- Provide the mechanism for solver transitions (i.e., from the default non-sequential modular solver to the simultaneous solver used by the Column or sub-flowsheet).

In addition, it is also possible to use the sub-flowsheet as a fluid package transition (i.e., switching from a fluid package tailored for VLE calculations to one tailored for LLE calculations), although, with UniSim Design 350 and higher, this is not the only mechanism for applying these transitions.

With other unit operations in UniSim Design, information can flow across the sub-flowsheet boundary bi-directionally (i.e., product stream information can flow into the sub-flowsheet).

Within a given flowsheet, all sub-flowsheets are treated as a single unit operation with multiple connections. The parent flowsheet (main or sub) in which that sub-flowsheet resides has no knowledge of what is inside the sub-flowsheet (i.e., it could be a refrigeration loop or a decanter system). From the parent flowsheet, the sub-flowsheet

behaves as any other operation and calculates whenever “feed” conditions change.

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 its owner flowsheet, you can, at any time, enter the sub-flowsheet to examine conditions in greater detail or make changes. You can make topology changes in the main PFD or in the sub-flowsheet environment. If you enter the sub-flowsheet’s build environment, the following UniSim Design behaviour occurs:

- The parent flowsheet (and all those which are above the current flowsheet in the simulation case hierarchy) are temporarily cached.
- The parent flowsheet’s solver(s) (and all those which are above the current flowsheet in the simulation case hierarchy) only process the forget pass, and calculations are temporarily suspended. Within the sub-flowsheet calculations are still performed, but the results are not propagated to the rest of the simulation until you come out of the sub-flowsheet environment. This lets you focus on a specific aspect of the simulation without having the entire simulation calculate every time conditions change.

While there are certain programmatic behaviours built into UniSim Design to facilitate the proper behaviour of the flowsheets, this does not limit its ability to access information from any location in the program. No matter where you are in the simulation case, you can open any flowsheet’s PFD, Workbook or property view for a stream or operation within that flowsheet. Since the sub-flowsheets are, in essence, single operations within the main flowsheet, each has its own property view. You can access resident information inside the sub-flowsheet through this property view without ever having to enter the sub-flowsheet itself.

The accessing of data within the simulation case is the function of the Navigators.

Refer to [Section 7.17 - Object Navigator](#) for details on the Object Navigator.

Refer to [Section 11.19 - Variable Navigator](#) for details on the Variable Navigator.

Refer to [Section 7.18 - Simulation Navigator](#) for details on the Simulation Navigator.

- The Object Navigator gives immediate access to the property view for any stream or operation from any location.
- The Variable Navigator lets you target variables from any Flowsheet for use—either by a logical unit operation—or as part of one of the Data Collectors.
- The Simulation Navigator provides a single location where you can view or interact with the property views for all streams and unit operations in the simulation case, regardless of which flowsheet they reside in.

3 Flowsheet

3.1 Introduction	2
3.2 Flowsheets in UniSim Design.....	2
3.3 UniSim Design Environments.....	4
3.3.1 Basis Environments	4
3.3.2 Simulation Environments	5
3.3.3 Environment Relationships	7
3.3.4 Advantages of Using Environments.....	8
3.4 Sub-Flowsheet Environment.....	10
3.4.1 Sub-Flowsheet Entities	10
3.4.2 Sub-Flowsheet Advantages.....	11
3.4.3 Multi-Level Flowsheet Architecture	11
3.4.4 Flowsheet Information Transfer	13
3.5 Templates.....	14
3.5.1 Template Information	15
3.5.2 Creating a Template Style Flowsheet.....	19
3.5.3 Installing a Template	20
3.6 Property View Flowsheet Analysis	22
3.6.1 Stream Analysis.....	22
3.6.2 Unit Operation Analysis.....	24

3.1 Introduction

The following sections describe the functionalities of the various flowsheets within UniSim Design.

Keep the following in mind:

- Fluid packages can be assigned to individual unit operations or groups of unit operations in a flowsheet, independent of the default fluid package for that flowsheet. Fluid package transitions are automatically introduced for the user.
- The Cut/Copy/Paste function allows the creation of an *.ufl file, which can be stored on disk. This UFL file can be created from any collection of unit operations and streams within the simulation case. The fluid packages associated with those objects are also saved with the UFL file. You can open this file and edit it, using the fluid packages contained for the calculations. When an UFL file is imported into a simulation case, the fluid packages are removed; only the objects, topology, and specifications are imported into the target case.
- Unit operations and streams can be added to and removed from flowsheets using the Cut/Copy/Paste functionality.
- Case Collaboration allows you to construct smaller flowsheets of a larger process and examine the relationships/impacts between flowsheets.
- XML data representation provides a complimentary representation of the traditional binary form of the case storage. It allows you to read impartial information into one or more existing simulation cases, resulting in those cases being updated with the new information.

In addition to these features, UniSim Design can define elements of the simulation case and store them independently of the case for subsequent re-use. This includes not only fluid package elements and flowsheet topologies, but analytical tools such as property calculations and Workbooks as well.

3.2 Flowsheets in UniSim Design

UniSim Design uses a multi-level flowsheet architecture tightly integrated within a framework of simulation environments. Separate Desktops for each environment help you focus on the current design task, and multi-level flowsheets allow you to contain complex processes within sub-flowsheets.

As a result, you can interact with an installed sub-flowsheet operation as if it were a simple black box, or you can use the sub-flowsheet's simulation environment when more detailed interaction is required.

Potentially complex flowsheets installed as sub-flowsheet operations function in a familiar and consistent manner, much like the other “normal” unit operations in UniSim Design.

UniSim Design also supports the concept of a Process Template. A template is a complete flowsheet that is stored on disk with additional information on how to set up the flowsheet as a sub-flowsheet operation.

Typically, templates represent a plant process module or a portion of a process module. The stored template can be read from disk and installed as a complete sub-flowsheet operation numerous times and in any number of different simulation cases.

The flowsheet and sub-flowsheet are not restricted to a single fluid package.

The sub-flowsheet can also be assigned a separate fluid package different from main flowsheet. This feature lets you model plant utilities more rigorously using, for example, cooling water and steam circuits as separate flowsheets with dedicated Steam Table property packages.

Column Sub-Flowsheets – A Special Case

Column sub-flowsheets are a distinct class of sub-flowsheets because they provide a simultaneous flowsheet solution. Even though they are different, they are created and accessed much like normal sub-flowsheets, and can be created and later imported into other simulations.

The column sub-flowsheet’s property view and the Column’s simulation environment are very different, however, 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 8 - Column](#) in the **UniSim Design Operations Guide**.

3.3 UniSim Design Environments

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

The environment design concept is one of the cornerstones on which UniSim Design is built. These environments let you 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 will not proceed with steady state calculations until you are finished working in the active area. Since the UniSim Design integrator is time-step based, the environments have no impact on dynamic calculations.

Separate Desktops are available within each environment. These Desktops include an appropriate menu bar, tool bar, and Home View(s) specifically designed for interaction with that particular environment. The Desktops also remember the views that are open, even when their associated environment is not currently active.

When moving from one environment to another, Desktops provide a mechanism for quickly and automatically **putting away** what ever views are open in one environment, and **bringing up** the views that were open in the other environment. This feature is useful when working with large flowsheets.

The environments in UniSim Design can be loosely grouped into two categories:

- Basis environments
- Simulation environments

These environments are described in more detail in the following sections.

3.3.1 Basis Environments

There are two types Basis environments:

- Simulation Basis
- Oil Characterization

Simulation Basis Environment

When beginning a UniSim Design simulation, you automatically start in the Simulation Basis environment. Here you 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 and library

and/or hypothetical components. Fluid packages can also contain information such as reactions and interaction parameters.

The Desktop for the Simulation Basis environment contains a tool bar with the appropriate icons for Basis tasks and designates the Simulation Basis Manager view as the Home View.

Oil Characterization Environment

The Oil Characterization environment lets you characterize petroleum fluids by creating and defining assays and blends. The Oil Characterization procedure generates petroleum hypocomponents for use in your fluid package(s). The Oil environment is accessible only within the Simulation Basis environment.

The Desktop for the Oil Characterization environment is very similar to the Desktop in the Simulation Basis environment. Icons specific to Generating Oils appear and the Oil Characterization Manager is the Home View.

3.3.2 Simulation Environments

The following are examples of Simulation environments:

- Main flowsheet environment/sub-flowsheet environment
- Column sub-flowsheet environment

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

UniSim Design allows you to “nest” flowsheets. The main flowsheet is the parent flowsheet for the sub-flowsheets it contains. A sub-flowsheet can also be a parent flowsheet if it contains other sub-flowsheets.

Main Flowsheet/Sub-Flowsheet Environment

The simulation case main flowsheet environment is where you do the majority of your work in UniSim Design. Here you install and define the following:

- Streams
- Unit operations
- Columns

You can also create sub-flowsheets in this main environment.

This flowsheet serves as the base level or “main” flowsheet for the whole simulation case. Any number of sub-flowsheets can be generated

in this main flowsheet. While there is only one main flowsheet environment, each individual sub-flowsheet that is installed can have its own corresponding sub-flowsheet environment.

The Desktop for the main flowsheet environment contains an extensive menu bar and tool bar designed for building and running simulations. There are two Home Views for the flowsheet: the Workbook and the PFD.

A sub-flowsheet environment is almost identical to the main flowsheet environment because you can install streams, operations, and other sub-flowsheets. One difference is that each installed flowsheet in the simulation case has 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 are put on hold until you return to the main flowsheet environment.



Parent Simulation Environment icon

The Desktop for a sub-flowsheet environment is virtually identical to the Desktop for the main flowsheet except for one difference: the Parent Simulation Environment icon appears in the tool bar.

Column Sub-Flowsheet Environment

The Column environment is where you install and define the streams and operations contained in a column sub-flowsheet, and it is similar to the sub-flowsheet environment described in the previous section.

Examples of unit operations you can install in a column sub-flowsheet include the following:

- Tray sections
- Condensers
- Reboilers
- Side strippers
- Heat exchangers
- Pumps

There are eleven pre-build columns available in UniSim Design.

UniSim Design contains a number of pre-built column sub-flowsheet templates that allow you to quickly install a column of a typical type and then, if necessary, customize it as required within its Column environment.



Column Runner icon

The menu bar, tool bar, and Home Views for the Column environment are designed expressly for designing, modifying, and converging column sub-flowsheets. It includes an additional Home View (the Column Runner), and a corresponding menu item and a Column Runner icon on the tool bar provide access to the Column Runner view. Even with these changes, a Column environment Desktop still closely

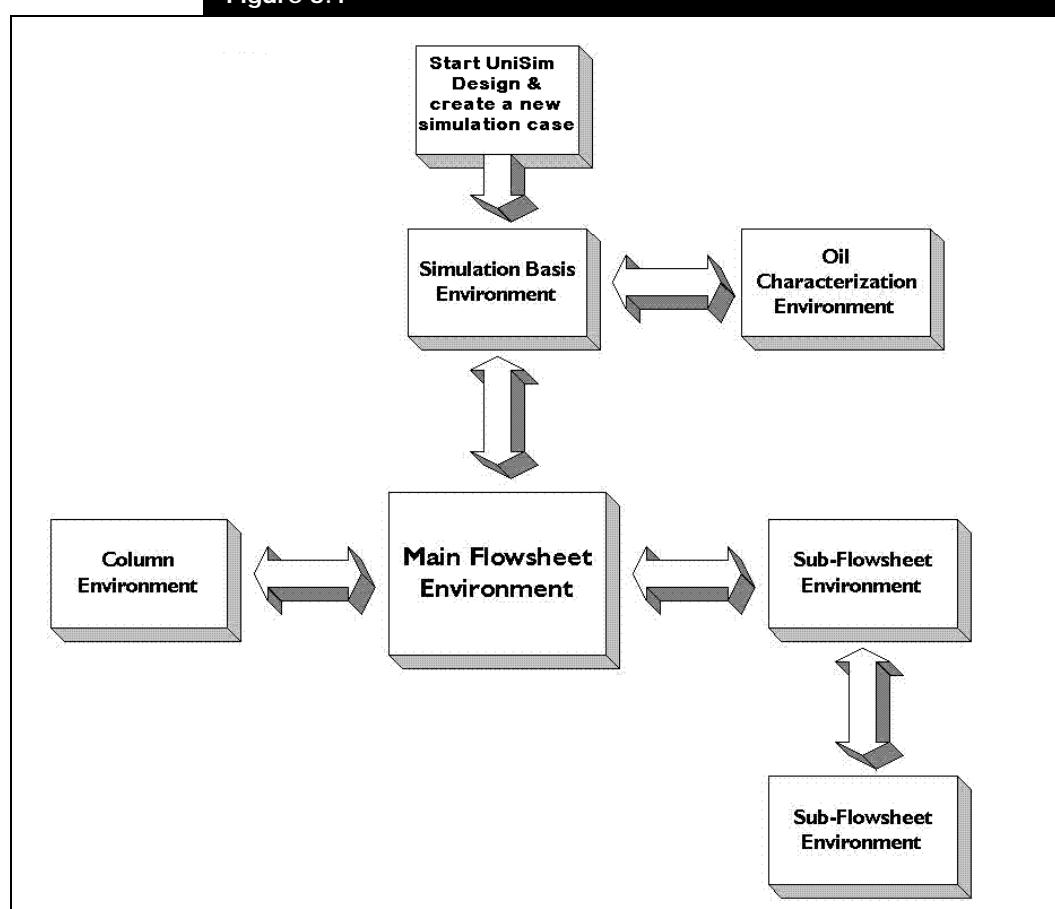
resembles the conventional flowsheet environment Desktop.

Due to the nature of its solution method, the column sub-flowsheet does not support other sub-flowsheets

3.3.3 Environment Relationships

The figure below shows the relationship among the various environments. The arrows indicate how you usually move between environments as you build a UniSim Design simulation.

Figure 3.1



Building a Simulation

1. Create a new simulation case. UniSim Design automatically starts you in the Simulation Basis environment.
2. Inside the Simulation Basis environment, do the following:

- Select a property method and pure components from the UniSim Design pure component library.
- Create and define any hypothetical components, if required.
- Define reactions, if required.

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.

To access the Oil environment you must be inside the Simulation Basis environment.

3. Enter the Oil Characterization environment, where you can do the following:
 - Define one or more Assays and Blends.
 - Generate petroleum hypocomponents representing the oil.
4. Return to the Simulation Basis environment.
5. Enter the main flowsheet environment, where you can do the following:
 - Install and define streams and unit operations.
 - Install columns operations, process templates, and sub-flowsheet operations as required.
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.

You can move between the flowsheet environments at any time during the simulation. The arrows in the previous 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.



Object Navigator icon

The Navigator lets you move directly from one flowsheet to any another. The only restriction is that the Oil environment can be accessed only within the Simulation Basis environment.

3.3.4 Advantages of Using Environments

Using environments helps 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 UniSim Design simulation case. When you start UniSim Design, you start in the Simulation Basis environment, where you define a fluid package by selecting a property method and components. When finished, you enter the main flowsheet environment and begin installing streams and unit operations.

If you are missing some components in the fluid package, you can return to the Simulation Basis environment and all flowsheets are placed in Holding mode until you return to the main flowsheet. This prevents calculations from occurring until you have made all required

changes to the fluid package.

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.

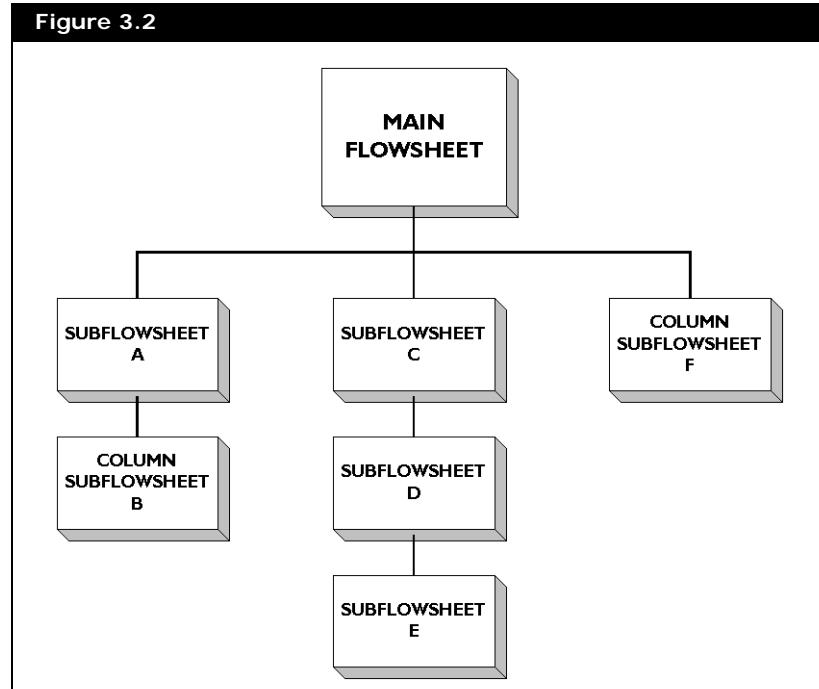


Click the Active icon to resume calculations.

The flowsheet calculations do not resume until you return to the main flowsheet environment.

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

Figure 3.2



You can change specifications from anywhere in the simulation case. You can make topology changes on any open PFD, or you can make the changes in the environment of that flowsheet.

If changing the number of trays for a column in sub-flowsheet F, enter the environment for this sub-flowsheet and make the changes. UniSim Design re-calculates the column. There are no flowsheets below F, so all other flowsheets are on hold while you modify the column.

Continue making changes until you reach a satisfactory solution for F, then return to the main flowsheet environment to automatically re-calculate all the flowsheets based on the new sub-flowsheet solution.

If you modify sub-flowsheet D, all flowsheets are on hold except D and E, which will solved based on your modifications. After reaching a new solution for D, enter sub-flowsheet C, which then resumes calculations. When you return to the main flowsheet, all other flowsheets (Main, A, B and F) resume calculations.

If you move directly from sub-flowsheet D to sub-flowsheet A, however, UniSim Design automatically **visits** the main flowsheet and updates all calculations, so flowsheet A has the most up-to-date information when you transfer to it. Any movement to a flowsheet not on your **branch** of the tree forces a full recalculation by UniSim Design.

3.4 Sub-Flowsheet Environment

See [Section 2.3 - Adding a Sub-Flowsheet](#) in the **UniSim Design Operations Guide** for more information about installing a sub-flowsheet.

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

The Simulation environment described in the previous section is one of the cornerstone design concepts upon which UniSim Design is built. When combined with sub-flowsheet capabilities, it defines the basic foundation for building a UniSim Design simulation. The sub-flowsheet and Column operations use the multi-level flowsheet architecture and provide a flexible, intuitive method for building 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 flowsheet, you can simulate each process unit inside its own sub-flowsheet.

3.4.1 Sub-Flowsheet Entities

Both the main flowsheet and sub-flowsheets contain 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, including unit operations, material and energy streams, utilities, etc.
A Dedicated PFD	A graphics view of the flowsheet showing the inter-connections between flowsheet objects.

Flowsheet Component	Description
A Dedicated Workbook	A tabular view describing the various types of flowsheet objects.
A Dedicated Desktop	The PFD and Workbook are home views for this Desktop, but also included are a menu bar and a tool bar specific to the specific flowsheet type.

3.4.2 Sub-Flowsheet Advantages

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

Once a template is installed, it is functionally equivalent to a sub-flowsheet that was created in that simulation case. It doesn't work the other way, however; you can't save a sub-flowsheet to disk and use it in another simulation.

The multi-flowsheet architecture of UniSim Design provides a number of technical and functional advantages. The following table explains the benefits of using sub-flowsheets in a simulation:

Capability	Benefit
Multiple Fluid Packages	Each installed sub-flowsheet can have its own fluid package within a single simulation case.
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, more easily managed components. This helps you to keep your simulation organized and concise.
Template Creation	Save time and money by creating individual template style flowsheets of commonly used process units, which you can install within other simulations. Templates are fully defined flowsheets with a property package and components, unit operations, streams, and flowsheet specifications.
Nested Flowsheets	You can use nested flowsheets, i.e., have sub-flowsheets inside other sub-flowsheets. The only restriction on nesting is you cannot create sub-flowsheet operations inside a Column sub-flowsheet.

The use of sub-flowsheets is the ideal solution if your simulation requires the use of multiple property packages or involves modeling large and complex processes.

3.4.3 Multi-Level Flowsheet Architecture

A Show/Hide command also exists for displaying sub-flowsheet objects on the main flowsheet PFD. For further details, see [Section 7.24.4 - Access Column or Sub-Flowsheet PFDs](#).

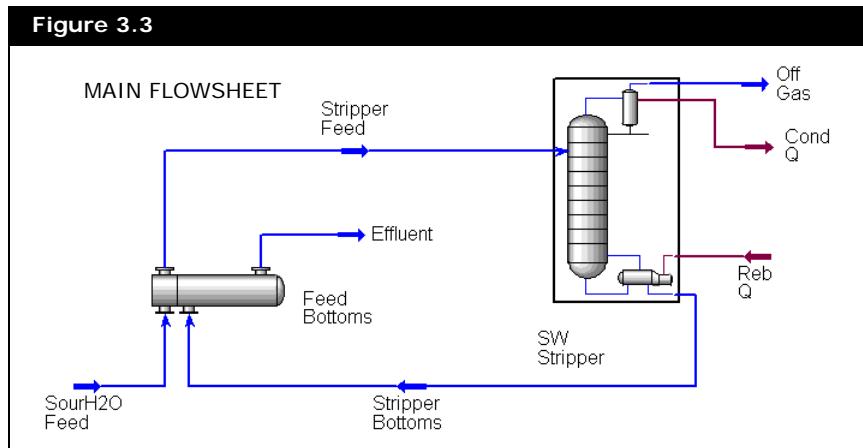
The sub-flowsheets contained in the main flowsheet of the simulation case are discrete unit operations with feed and product streams. If you are interested only in the feeds to and the products from a sub-flowsheet, you can work from the main flowsheet.

If you need to view information about the individual operations in the

sub-flowsheet, go “inside” the sub-flowsheet to get a more detailed perspective. This is also referred to as “Entering the sub-flowsheet environment”.

This concept also applies to column operations. For example, consider the PFD of the main flowsheet for the Sour Water Stripper simulation shown below.

In the main flowsheet, the column appears as any other unit operation ([Figure 3.3](#)), however, the column has its own sub-flowsheet ([Figure 3.4](#)) that provides a detailed look at the column’s internal streams and operations.



From the simulation environment of the main flowsheet, the distillation column SW Stripper appears as any other unit operation with feed and product streams, however, the column is also a sub-flowsheet with streams and operations of its own.

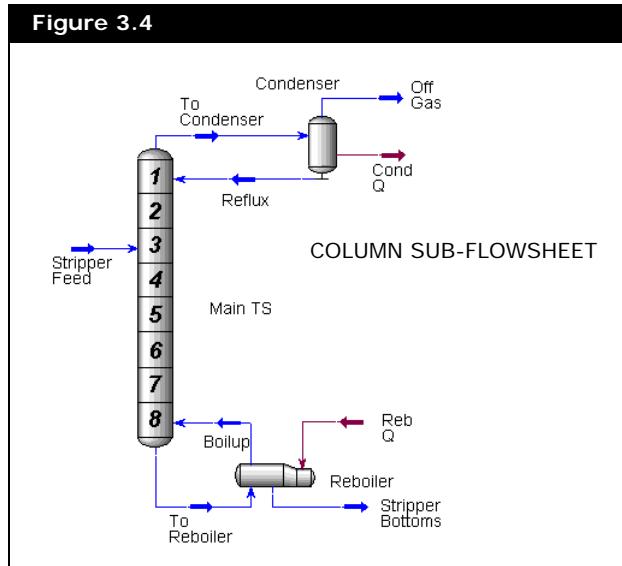
For a more detailed look at the column, go **inside** the column sub-flowsheet and examine the streams and operations in the SW Stripper’s simulation environment. Inside the column sub-flowsheet (see [Figure 3.4](#)), the tray section, reboiler, and condenser exist as individual unit operations. Similarly, the streams attaching these operations are also distinct.

Within the main flowsheet, the only sub-flowsheet streams of interest are those that directly attach to the column. In the case of the Sour Water Stripper, the material streams Feed, Off Gas, Bottoms, and the utility streams Cooling Water and Steam are the streams of interest. These streams are called 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, which display only the information related to that flowsheet.

Within the sub-flowsheet environment, a dedicated Workbook and PFD allow you to access to the information that pertains only to this sub-flowsheet. Although information is never hidden or inaccessible among the various levels of flowsheets in a simulation case, the use of the environments organizes and focuses the simulation efforts in a clear

and logical manner.



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

Multi-Flowsheet Navigation

The multi-flowsheet architecture 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 becomes the contents of the files.

Refer to [Section 7.18 - Simulation Navigator](#) for details on the Simulation Navigator.

Refer to [Section 7.17 - Object Navigator](#) for details on the Object Navigator.

UniSim Design has special tools called Navigators that are designed to take advantage of this directory-like structure. Within a single view, you can easily access a stream, operation, or process variable in one flowsheet from any other flowsheet in your simulation.

3.4.4 Flowsheet Information Transfer

When you install or create a sub-flowsheet in the Simulation environment, it appears and behaves as a single operation with one or more feed and product streams. Whenever the values of the streams attached to the sub-flowsheet change, the sub-flowsheet recalculates

just like any other regular unit operation.

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

Each of the parent flowsheet's streams attached to the sub-flowsheet as either a feed or product are 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.

When a connection is established across the boundary, the sub-flowsheet is automatically renamed 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 do not require the same name. For example, you can have a stream named **To Decanter** in the main flowsheet connected with the **Decanter Feed** stream in a sub-flowsheet.

The sub-flowsheet architecture allows the consistent use of different property methods. On each sub-flowsheet's property view, UniSim Design allows you to control how stream information is exchanged as it crosses the flowsheet boundary.

Component maps are available to allow you to define how to handle different component lists between fluid packages.

For example, you can specify the Vapour Fraction and Temperature (specified or calculated values) of a stream in the Main simulation to be passed to the sub-flowsheet. Once this information is passed to the sub-flowsheet, the property package for the sub-flowsheet then calculates the remaining properties using the transferred composition.

No flash calculations are required for Energy streams. The heat flow is simply passed between flowsheets.

3.5 Templates

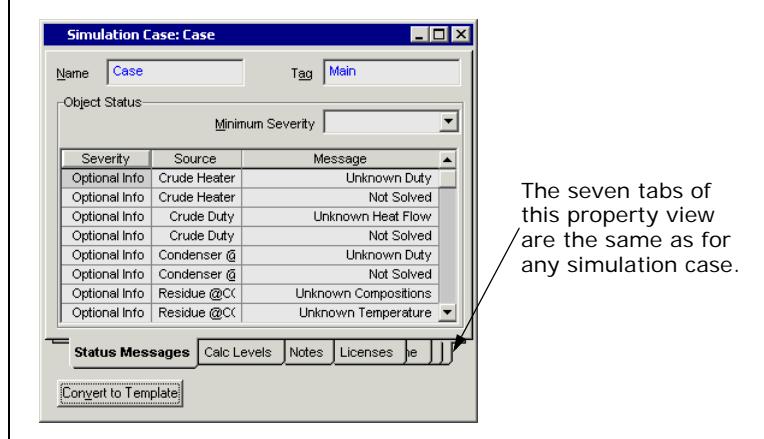
A template flowsheet is a normal UniSim Design flowsheet with some additional information contained in its main properties. It uses a different file extension when it is stored to disk (*.tpl or *.ufl instead of the regular *.usc). The different file extension is used mainly for organizational purposes.

3.5.1 Template Information

To open the Simulation Case view, select Main Properties from the Simulation menu, or press **CTRL M**.

The template information for the flowsheet is accessed through the Simulation Case view.

Figure 3.5



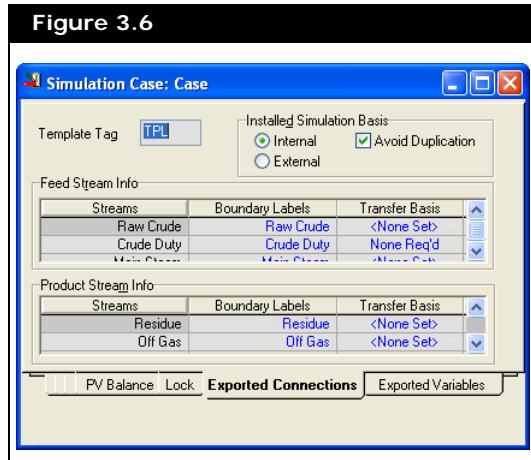
There are two additional tabs that are exclusive to templates, which are available when the standard simulation case is converted to a template. To convert a simulation case to a template, click the **Convert to Template** button at the bottom of the Simulation Case view. Once you click the button, the extra tabs appear and the button is no longer visible.

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" that you can install as a sub-flowsheet operation with the same ease and in the same manner as you would install a regular unit operation.

Exported Connections Tab

On the Exported Connections tab, enter the Template Tag and select the Installed Simulation Basis. All Feed and Product connections also

appear.



Template Tag

Flowsheet Tags are short names used by UniSim Design 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 **TPL** (for template). When more than one sub-flowsheet operation is installed, UniSim Design ensures unique tag names by adding an incremental numerical suffix similar to the UniSim Design 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

When a template is imported into a simulation case, its associated fluid package is added to the list of fluid packages in the Simulation Basis Manager view. 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 where it is installed. This only affects what happens at the time the template is first installed.

Once a template is installed the resulting fluid package association can be over-ridden in the Simulation Basis Manager view at any time.

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 appear in the appropriate group. Boundary Streams cannot be selected to appear on this tab; they are automatically determined by UniSim Design. These are the streams that you are 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 describes 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 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 assumes the name of its corresponding boundary stream in the template.

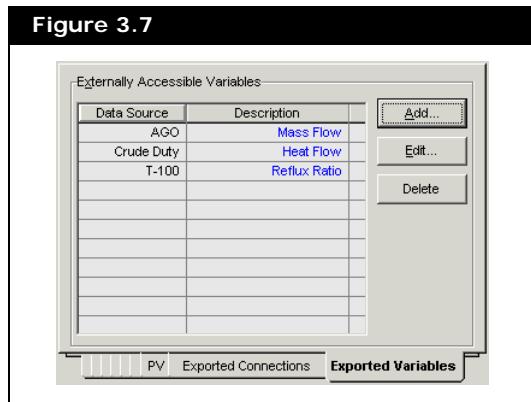
The Transfer Basis is used for feed and product streams as they cross the flowsheet boundary. The Transfer Basis becomes significant only when the sub-flowsheet and parent flowsheet property packages are different. When there are differing fluid packages in the two flowsheets (parent and sub-flowsheet) you can specify what stream properties are used to calculate the stream on the other side of the boundary.

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

Exported Variables Tab

Use this tab to create and maintain the list of Exported Variables.



Although any information can be accessed inside the sub-flowsheet using the Variable Navigator, this feature can target key process variables inside the sub-flowsheet and display their values 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 useful when treating the sub-flowsheet as a "black box" as all the important specifications for the operation of the sub-flowsheet can be collected and documented in one location. You will not have to enter the sub-flowsheet environment to adjust the template to your needs.

To add variables to this tab, click the **Add** button. The Add Variable to Case view appears. Select the flowsheet object and variable. On the Add Variable to Case view, you can override the default variable description and provide another one.

Refer to [Section 2.2 - Sub-Flowsheet Property View](#) in the [UniSim Design Operations Guide](#) for information about the Parameters tab.

When installing a template into another case, these variables appear on the Parameters tab of the sub-flowsheet property view.

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 just get a warning message and the extra information is ignored.

3.5.2 Creating a Template Style Flowsheet

Any main flowsheet can be used as the base for a template. By pressing a button, you toggle the flowsheet so that it becomes a template style flowsheet. Then you supply the extra information needed for installing in any simulation case. You can also save it to disk.

Once you convert a case into a template and import it into another case, most of the monitoring and customizing tools within the template will be transferred over to the new simulation environment. Information such as the strip charts, utilities, and macro language entries are transferred along with the template. However, the event schedules, optimizer settings, dynamics initialization settings, snapshot manager settings, and Databook items (for example, case studies, process data table, and data recorder settings) are not transferable.

You cannot create a template from just parts of a main flowsheet. Delete any unwanted streams and operations from the main flowsheet before saving it to disk. To preserve your original simulation case, save it with another name before creating the template.

UFL files, which are created using the Cut/Copy and Paste functionality (accessible from the PFD), have near equivalent behaviour to template files and can be created by selecting groups of operations from within a simulation case.

You can convert an existing case to a template or create a new template for a flowsheet.

Converting an Existing Case to a Template

You can convert an existing case to a template if you have already created a simulation case (not a new template), or if you have an existing case on disk that you want to use as a template.

1. From the **Simulation** menu, select the **Main Properties**. The Simulation Case property view appears.
2. Click the **Convert to Template** button.
3. Click on the **Exported Connections** tab.
4. Set the Template Tag, Installed Simulation Basis and other optional template information if required.

Refer to [Section 3.5.1 - Template Information](#) for more information.

5. When you save the simulation, it is saved as a template.

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

Creating a New Template

You can create a new template for a new simulation case.

1. From the **File** menu, select the **New** command, and then **Template**.
2. Build the simulation.
3. From the **Simulation** menu, select **Main Properties**. The Simulation Case property view appears.
4. Click on the **Exported Connections** tab.
5. Set the Template Tag, Installed Simulation Basis and other optional template information if required.
6. When you save the simulation, it is saved as a template.

Since you can have multiple simulation cases in memory, you can create a new template as part of your current session and then install it in your original Simulation case.

UniSim Design automatically saves the template in the Templates directory as a template file (*.tpl). The default path for the Templates directory is set according to the UniSim Design preferences. As shipped, the default directory is **UniSim Design\Template**.

A combination of flowsheets can be in your template (i.e., a main flowsheet and one or more sub-flowsheets). Likewise, multiple fluid packages can be included in the template if they are associated with a flowsheet at the time the template is saved to disk.

3.5.3 Installing a Template

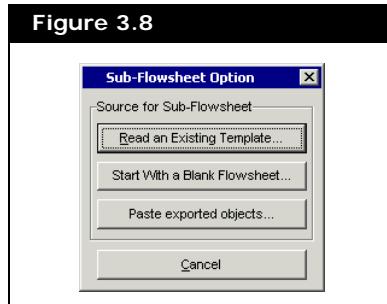
To install a template, follow the same procedure as for installing a sub-flowsheet. Do **one** of the following:



Sub-flowsheet icon

- Select the **Flowsheet-Add Operation** command (or press **F12**), to open the UnitOps view. Select **Standard Sub-Flowsheet** from the Available Unit Operations list.
- Open the UniSim Design Object Palette (press **F4**) and click the **Sub-Flowsheet** icon on the Object Palette.

After you initiate the installation of a sub-flowsheet, a Sub-Flowsheet Option view appears.



Click the Read an Existing Template button to install a template from disk.

If you do not want to read an existing template, click the Start with a Blank Flowsheet button.

The Start with a Blank Flowsheet is a good option if you are just creating a small sub-flowsheet. This sub-flowsheet is not available for use in any other simulation case you create in the future.

If you think you might re-use the sub-flowsheet at a later date, consider creating a full template flowsheet instead.

The process of creating a sub-flowsheet with a blank initial flowsheet operation is covered in detail in [Section 2.3 - Adding a Sub-Flowsheet](#) of the [UniSim Design Operations Guide](#).

Reading an Existing Template

UniSim Design includes a number of sample process template for trial purposes.

To install a template style flowsheet, click the Read an Existing Template button. UniSim Design looks in the default Templates directory [**Root:\UniSim Design\Template**] for available template files (*.TPL).

UniSim Design automatically installs any fluid packages associated with 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.

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

After the flowsheet(s) have been inserted in the simulation case, a fluid package is selected for the sub-flowsheet based on the Installed Fluid Package setting used in the template.

Once UniSim Design finishes installing the template, you are placed on the Connections tab of the sub-flowsheet property view where you can define the connections for the template.

3.6 Property View Flowsheet Analysis

In UniSim Design, stream and operation property views contain analytical information based on the current flowsheet conditions. For example, the stream property view has a page that contains information concerning all phases present in the stream. Also, certain operations have pages that display performance profiles, results, and other analytical information.

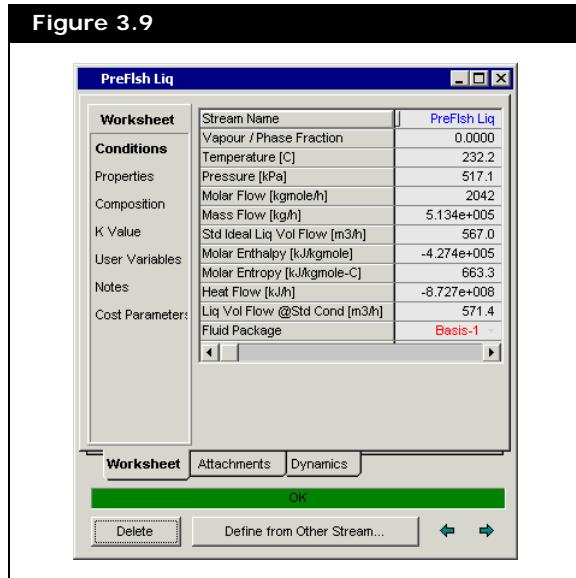
3.6.1 Stream Analysis

Refer to [Section 3.1 - Material Stream Property View](#) in the [UniSim Design Operations Guide](#) for details on the various tabs and pages.

See [Section 7.26 - Utilities](#) for information on attaching a utility to a stream.

The stream property view has two tabs that contain information

pertinent to stream analysis: the Worksheet and Attachments tabs.



The Properties page in the Worksheet tab contains detailed property correlation information about the stream. The Conditions page is 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 which unit operations are attached to the stream.

To view the stream properties, do the following:

1. Open the Stream view, click the **Worksheet** tab, then select the **Properties** page.
2. Click the **View Correlation Set List** icon. The Correlation Set Picker view appears.
3. Select **Standard Set** from the view and click the **Apply** button.



View Correlation Set List icon

The Correlation Set Picker view closes and the stream properties appear in the table on the Properties page. In addition to containing the basic stream conditions, more detailed physical property information for the stream is shown in the table.

With the stream view at its default size, the page has horizontal scroll bars. 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 also resize it so that all phases, and all of the properties for each phase can be seen, as shown

in the figure below.

Figure 3.10

Worksheet	Stream Name	PreFlsh Liq	Vapour Phase	Liquid Phase
Conditions	Vapour / Phase Fraction	0.0000	0.0000	1.0000
	Temperature [C]	232.2	232.2	232.2
	Pressure [kPa]	517.1	517.1	517.1
	Molar Flow [kgmole/h]	2042	0.0000	2042
	Mass Flow [kg/h]	5.134e+005	0.0000	5.134e+005
	Std Ideal Liq Vol Flow [m3/h]	567.0	0.0000	567.0
	Molar Enthalpy [kJ/kgmole]	-4.274e+005	-1.292e+005	-4.274e+005
	Molar Entropy [kJ/kgmole-C]	663.3	287.2	663.3
	Heat Flow [kJ/h]	-8.727e+008	0.0000	-8.727e+008
	Liq Vol Flow @Std Cond [m3/h]	571.4	0.0000	571.4
	Fluid Package	Basis-1		

Worksheet Attachments Dynamics

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.

3.6.2 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 UniSim Design have pages that contain analytical information.

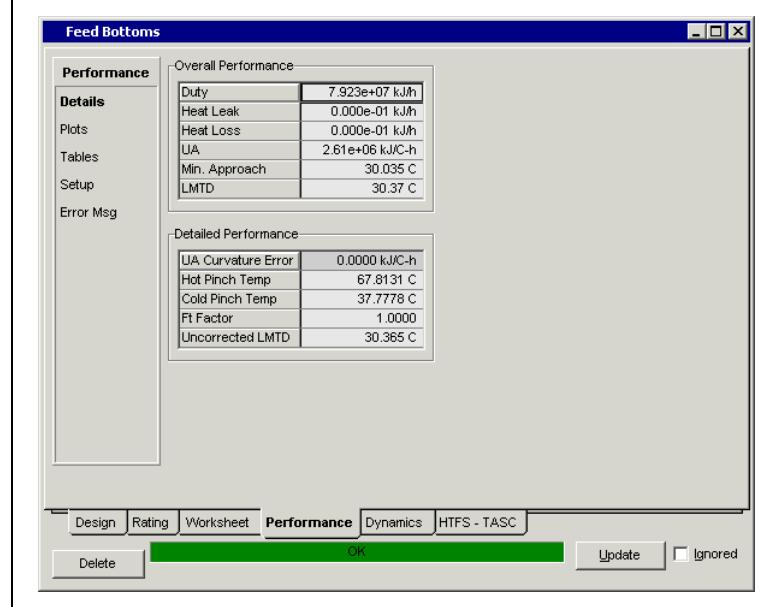
The type of analytical information found in operation property views depends on the operation type. Regardless of what the operation is, the displayed information is automatically updated as conditions change.

For example, the Heat Exchanger displays its analytical information on the Worksheet and Performance tabs.

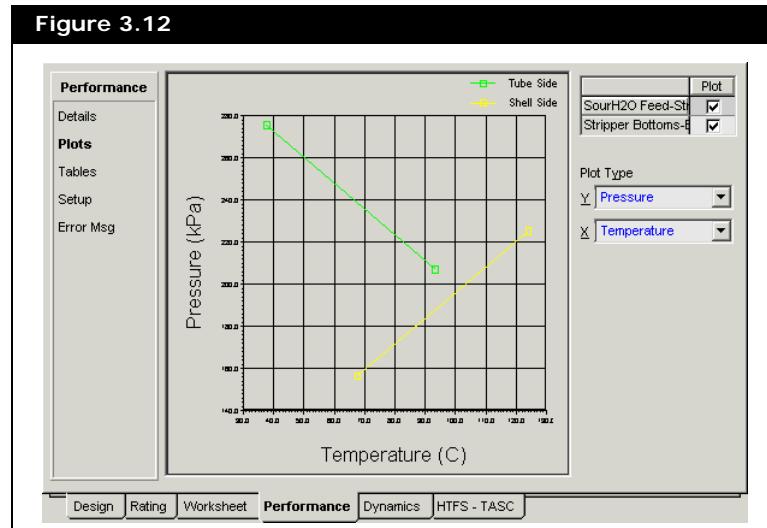
The Details page in the Performance tab of a Heat Exchanger displays the information in two groups:

- Overall Performance

- Detailed Performance

Figure 3.11

The Plots page lets you generate curves for the shell and/or tube sides of the heat exchanger. From the Plot Type drop-down list, you can select the X and Y axis variables for the plot.

Figure 3.12

The Tables page displays the same information provided in the Plots page but in tabular form.

4 File Management

4.1 Menu Bar	2
4.2 File	2
4.2.1 New and Open Commands	3
4.2.2 Saving Commands	6
4.2.3 File Extensions	9
4.2.4 Closing Commands.....	10
4.2.5 Printing	11
4.2.6 Exiting UniSim Design	11
4.3 UFL Files	11

4.1 Menu Bar

Most of the tasks performed in UniSim Design are accessed through the menu bar. The list of command or function groups, displayed at the top of the UniSim Design Desktop, is a drop-down menu system. By selecting an option from the menu bar, a menu of commands appears.

The menus available in the menu bar changes depending on the simulation environment. For example, the Column environment has a menu item called Column in the menu bar.

If you want to switch focus from the menu bar without making a selection, press the **ESC** key or the **ALT** key.

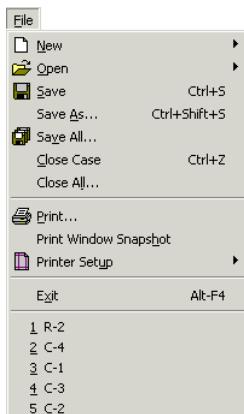
The menu bar also provides access to functions that can only be accessed this way, such as Session Preferences view (units, default naming schemes, etc.) and switching to another simulation currently in memory.

You can access the menu bar options in three ways:

- Click the required menu to open the associated drop-down menu.
- Use the **ALT** key in combination with the underlined letter in the menu bar title. For example, **ALT F** opens the **File** 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 UniSim Design, you can manoeuvre through the menu 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, opening the associated drop-down menu.

4.2 File

There are two variations of the File menu. A condensed menu appears in UniSim Design before a simulation is created or opened. The commands common to both versions of the menu, as well as the functions specific to the detailed menu, are explained in this section.



File Menu

The menu commands are grouped into five main categories:

- Starting a Simulation
- Saving a Simulation
- Closing a Simulation
- Printing
- Exiting UniSim Design

A command item with an arrow head pointing to the side contains additional commands in a submenu.

4.2.1 New and Open Commands

The New and Open commands under File in the menu bar enable you to create a new file or open an existing file. The file type can either be a type of simulation case (flowsheet) or case scenario manager project. After you select New or Open, an expandable menu appears showing the available commands:

Option	Description
Case	Creates a new simulation case or opens an existing one. This command enables you to access UniSim Design simulation cases (*.usc), Legacy HYSIM simulation cases (*.sim) or Backup simulation cases (*.bk0).
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.
Cut/Copy/ Paste	Creates a new blank case and then imports the selected UFL file into that case. Refer to Section 4.3 - UFL Files for more information regarding the UFL files.
Case Scenario Project	Create a new case scenario project or open an existing one.

When opening a case from an older version of UniSim Design,

you receive the following message.

Figure 4.1

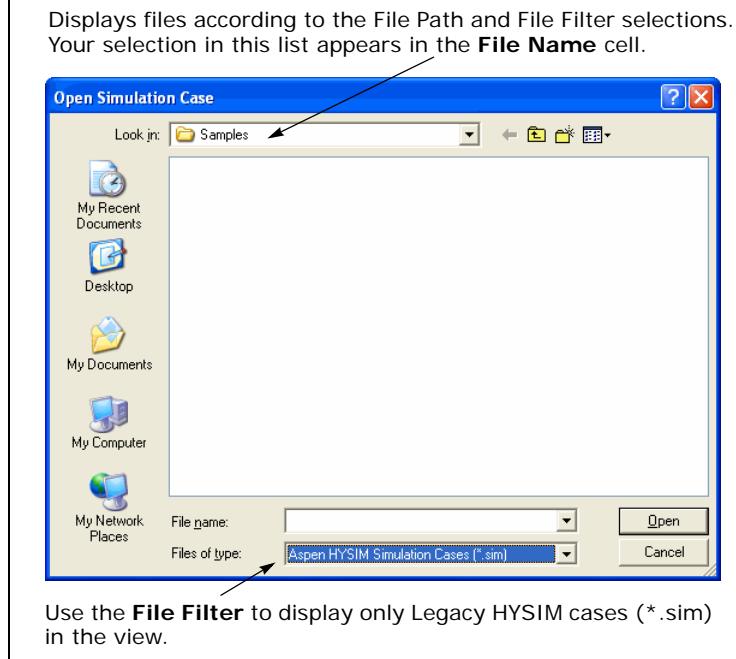


It is not necessary to go into the Simulation Basis environment. The warning lets you know that some objects will be recalculated when the case is loaded.

Reading a Legacy HYSIM Case

UniSim Design presents the functionality to open Legacy HYSIM simulations and to transfer all compatible data into the appropriate UniSim Design environments.

Figure 4.2



If the **Custom file picker** radio button is selected in the Session Preferences view, then the Open and Close view displays the

description for any Legacy HYSIM case with a Revision number.

If the value shown in the revision number is less than 10014, the Legacy HYSIM case is not valid for transfer into UniSim Design.

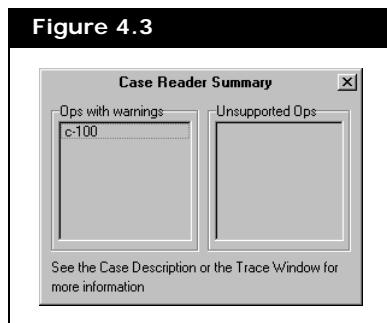
To open a Legacy HYSIM case:



Open Case icon

1. From the **File** menu in the menu bar, click **Open**, then click **Case** from the submenu, or click the **Open Case** icon.
2. On the Open Simulation Case view, select **Legacy HYSIM Simulation Cases (*.sim)** from the File Filter drop-down list.
3. Select a directory that contains a Legacy HYSIM case in the File Path group.
4. Select a Legacy HYSIM case in the list of cases or type the name of a case in the **File Name** input cell.
5. Click the **OK** button.
6. As UniSim Design reads a Legacy HYSIM case, the simulation is rebuilt in a piece-by-piece fashion. If an incompatibility is encountered, a message is recorded in both the Trace Window and the Case Description.

At the end of the case recall procedure, UniSim Design displays a summary on the Legacy HYSIM Case Reader Summary view.



The messages are separated into two groups:

- Ops with warnings
 - Unsupported Ops
7. You can view a summary list of messages by doing either of the following:
 - Scroll through the messages in the Trace Window.
 - Click the **Main Properties** command in the **Simulation** menu. The Main Properties view appears. Click the **Status Messages** tab.

Refer to [Section 1.3 - Object Status Window/Trace Window](#) for details on the Trace Window.

Legacy HYSIM Functionality Not Supported in UniSim Design

UniSim Design does not currently support Legacy HYSIM functionality, so it is not possible to transfer all Legacy HYSIM information into UniSim Design. The following table lists some of the issues:

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. UniSim Design supports only one diameter, one weir length and one weir height per Tray Section. In this case, dimensions of the 2 nd stage from the bottom of the Legacy HYSIM column are used for the UniSim Design Tray Section.
Liquid Extractor	Pump Arounds
Cyclone	Liquid Streams
Hydrocyclone	Vapour Streams
Rotary Vacuum Filter	Only Connections are transferred. Other operation parameters must be specified in UniSim Design.
Baghouse Filter	Liquid Streams
Crystallizer Solid Operation	All
Tee	Energy Stream Attachments
LNG	If the LNG Duty Stream is attached to another operation in Legacy HYSIM, the flowsheet is not complete in UniSim Design.
Plug Flow Reactor (PFR)	Space Time Option
CSTR	Space Time Option
CSTR	Dead Space Option
CSTR	Initialization from Stream
Data Recorder	Records variables while a flowsheet is being converged when adjusts or recycles are present.
Electrolyte	Legacy HYSIM functionality is not supported at all for simulation cases with Electrolytes OLI property package.

4.2.2 Saving Commands

If the active file is a simulation case, UniSim Design has the following

save commands:

Command	Description
Save	Saves the case using the current file name and location.
Save As	With this save command, you can enter a name and location to save the file. When you select the Save As command, either the "Revision Control on Save As" view, or the "Save Simulation Case As" view will appear, depending on if the "Show a pop-up view on Save As" checkbox is checked or not on the Preferences/Files/Revision Control page. You are able to select the File Path and a File Name for the case. UniSim Design automatically attaches the default file extension, *.usc. You can also save the entire case as an XML file or an UFL file.
Save All	Use this command to save all currently opened UniSim Design files. You are asked to select which files should be saved. You can select more than one file at a time by holding down the CTRL key then clicking each file you want to select. Click Save to save the files as shown in the view, or Save As to save with a new name and/or location.

Figure 4.4

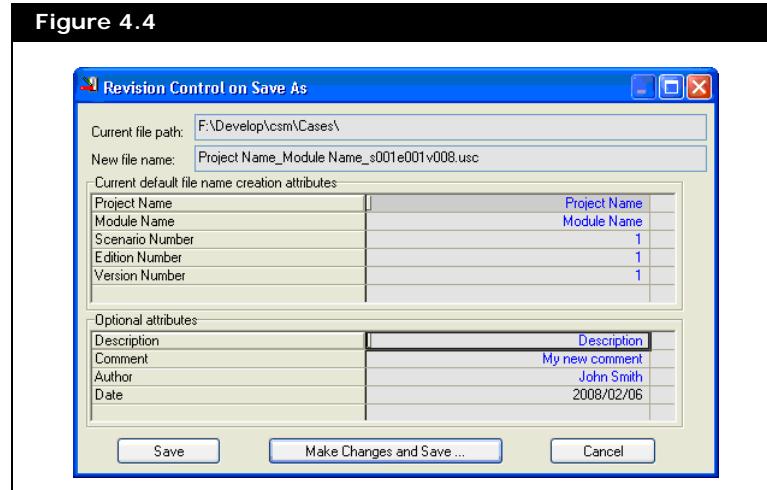


Figure 4.5

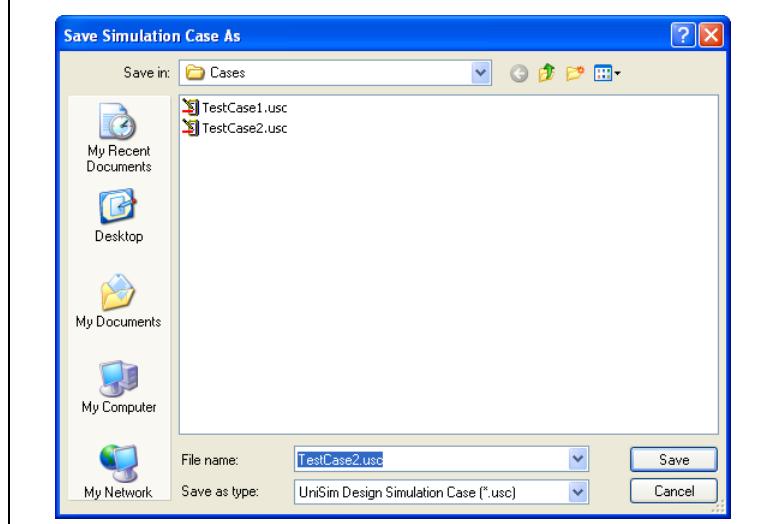
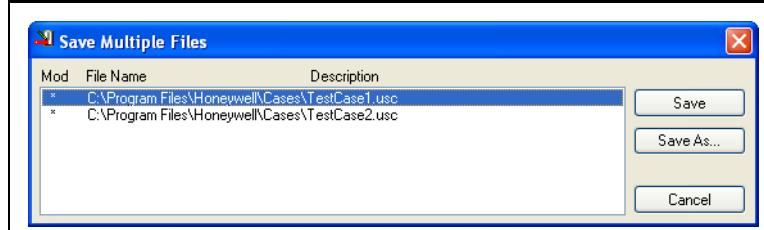


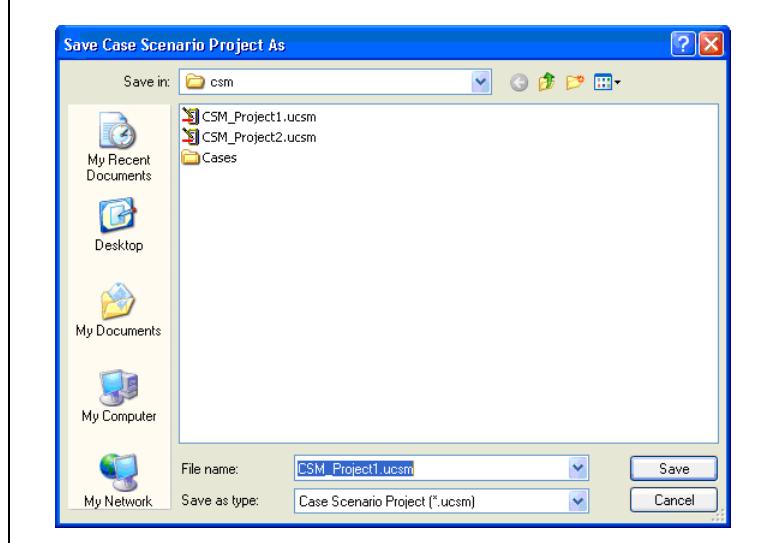
Figure 4.6



If the active file is a case scenario project, UniSim Design has the following save commands:

Command	Description
Save	Saves the case scenario project using the current file name and location.
Save As	When you select the Save As command, the "Save Case Scenario Project As" view will appear, allowing you to select a different location and file name. UniSim Design automatically attaches the default file extension, *.ucsm.
Save All	Use this command to save all currently opened UniSim Design files. You are asked to select which files should be saved. You can select more than one file at a time by holding down the CTRL key then clicking each file you want to select. Click Save to save the files as shown in the view, or Save As to save with a new name and/or location.

Figure 4.7



4.2.3 File Extensions

The following table contains the file extensions that are available in UniSim Design:

File Extension	Description
.	All Files
*.bk?	Backup Simulation Cases
*.cml	Component Lists
*.col	Column Templates
*.csv	Historical Data CSV History Files
*.dll	Application Extensions (DLLs)
*.DMP	Historical Data DMP History Files
*.EDF	Extension Definition Files (EDFs)
*.fpk	Fluid Packages
*.hvv	User Variables
*.hyp	Hypothetical Groups
*.inp	PRO II to UniSim Design
*.oil	Oil Assay Files
*.PRF	Preference Files
*.rst	Reaction Sets
*.sch	Schedule Files
*.scp	Script Files
*.sim	Legacy HYSIM Simulation Cases
*.tpl	Template Cases
*.ufl	UniSim Design UFL Files
*.usp	UniSim Design USP Cases
*.usc	UniSim Design Simulation Cases

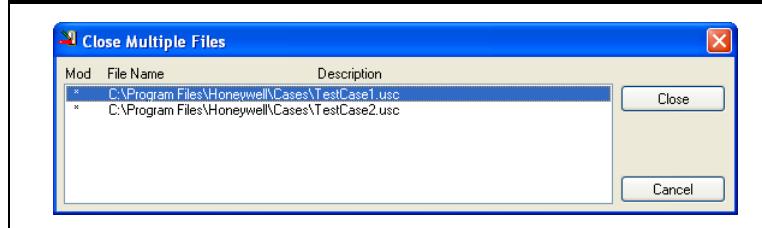
File Extension	Description
*.ucsm	UniSim Design Case Scenario Projects
*.uscv	UniSim Design Selected Variables Files for Case Scenario Manager
*.wrk	Workbook Files
*.WWB	WinWrap Basic
*.XML	UniSim Design XML Cases
*.xml	HYCON XML to UniSim Design

4.2.4 Closing Commands

If the active file is a simulation case, UniSim Design has the following commands for closing files:

Command	Description
Close Case	Closes the active case. Before closing the case, you are asked if you want to save the case.
Close All	Allows you to close more than one file at a time. The name of each opened file appears in the Close Multiple Files view. You select which files you want to close.

Figure 4.8



If the active file is a case scenario project, UniSim Design has the following commands for closing files:

Command	Description
Close Case Scenario Project	Closes the active case scenario project. Before closing the file, you are asked if you want to save the file if it has been modified.
Close All	Allows you to close more than one file at a time. The name of each opened file appears in the Close Multiple Files view. You select which files you want to close.

4.2.5 Printing

See [Section 9.2 - Printing in UniSim Design](#) for more information.

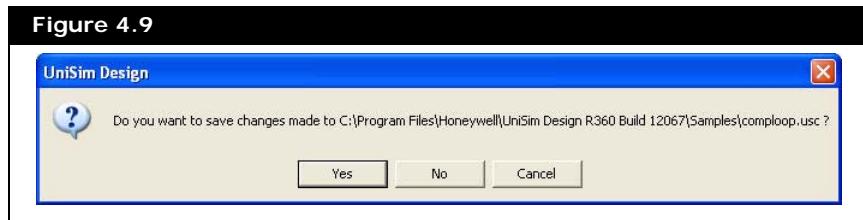
UniSim Design has the following print commands:

Command	Description
Print	Allows you to print Datasheets for streams and operations.
Print Snapshot	Prints a bitmap snapshot of what currently appears in the active UniSim Design 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.

4.2.6 Exiting UniSim Design

You can close UniSim Design by opening the File menu and clicking the Exit command.

If you have not saved your case before you select the Exit command, a warning message appears prompting you to save the case before exiting the program.



- If you want to save the case and exit UniSim Design, click the **Yes** button.
- If you do not want to save the case and still exit UniSim Design, click the **No** button.
- If you do not want to exit the UniSim Design program, click the **Cancel** button to stop the exit command.

4.3 UFL Files

The Object Inspect menu from the PFD contains a Cut/Paste Objects command. The sub-commands under the Cut/Paste Objects command allows you to copy, clone, cut, paste, export (copy to file), and import (copy from file) objects. When these commands are used, information about the objects is stored in an UFL file.

Information can be pasted into a different case or copy of UniSim Design.

- When the Copy, Cut or Clone command is used, a temporary UFL file is created in the default temporary directory. This file gets overwritten each time one of these commands is used by

any copy of UniSim Design that is running. When the paste command is used, this UFL file is used to import the objects back into UniSim Design.

- When the “Copy Object to File” or “Paste Object from File” command is used, the user can explicitly provide a name for the UFL file that is used. By default the file is saved in the UniSim Design template directory. You can access this file from a different case, or send this file to other UniSim Design users.

The UFL files are not full simulation case files, and do not contain case information on the Databook, strip charts, utilities, Optimizer, DCS driver, or event scheduler.

The entire sub-flowsheet can also be readily copied.

UFL files contain information about the objects that have been copied, so when importing or exporting UFL files, you get all the information required to restore the objects in a case (including fluid package information), but not the entire flowsheet or sub-flowsheet the object resides in.

UFL files generally contain a piece of a case that has been copied, and are typically imported into an existing open case.

You can use the Save As command from the File menu to save an entire case as an UFL file. This option saves you the time and trouble of selecting every object within the main PFD and exporting them to an UFL file. For convenience, UniSim Design also allows you to open an UFL file directly rather than importing it into an existing open case. But in this situation, a blank new case is created and the UFL file is actually imported into the new case.

UFL files are a bit more flexible than templates (*.tpl files). When you create a new template, you have to convert an entire case into a template and when that template file is read into a UniSim Design case, it always becomes a new sub-flowsheet. Each time you convert the case to a template, you gain one level of flowsheets and there is no way to move objects to different flowsheets.

The cut/copy/paste commands are easier to use because you can quickly operate on a selected group of objects only and paste objects back into any existing flowsheet without always creating a new sub-flowsheet.

5 Basis Environment

5.1 Introduction	2
5.2 Simulation Basis Manager.....	2
5.2.1 Components Tab	3
5.2.2 Fluid Packages Tab	10
5.2.3 Hypotheticals Tab.....	14
5.2.4 Oil Manager Tab.....	16
5.2.5 Reactions Tab	17
5.2.6 Component Maps Tab	21
5.2.7 User Property Tab	22
5.3 Reaction Package	25
5.4 Component Property View	26
5.4.1 Viewing a Pure Component.....	27
5.4.2 Defining a Traditional Hypothetical Component.....	27
5.4.3 Defining a Solid Hypothetical Component	28

5.1 Introduction

The Basis environment contains the following environments:

- Simulation Basis environment
- Oil Characterization environment

Refer to [Chapter 6 - Oil Characterization Environment](#) for more information.

Simulation Basis Environment

When beginning a new simulation case, UniSim Design automatically starts you in the Simulation Basis environment where you can create, define and modify fluid packages for use by the simulation flowsheets. In general, a fluid package contains a minimum of one property package and library and/or hypothetical components. Fluid packages can also contain information for reactions and interaction parameters.

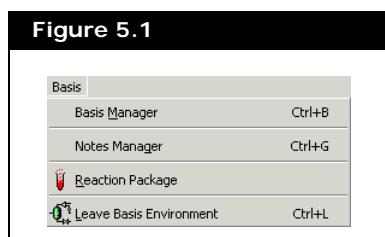


Basis icon

You can re-enter the Simulation Basis environment from any flowsheet by selecting the Enter Basis Environment command in the Simulation menu, or clicking the Basis icon found in the tool bar of both the Main and Column environments. For more information about the Basis environment, refer to the [UniSim Design Simulation Basis Guide](#).

Basis Menu

The Basis menu appears in the menu bar when you enter the Basis environment. The options available in this menu appear in the following figure.



5.2 Simulation Basis Manager

The Simulation Basis Manager view allows you to create and manipulate every fluid package in the simulation. Each flowsheet in UniSim Design can have its own fluid package.

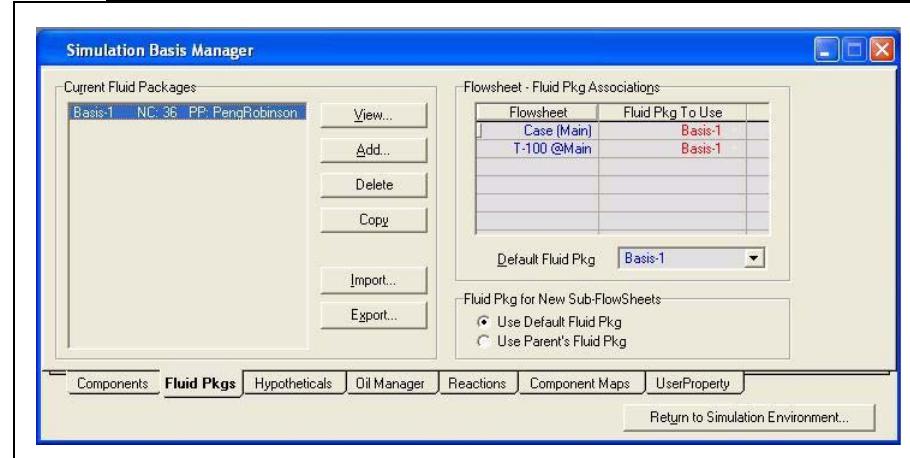
The template and column sub-flowsheets reside inside the Main Simulation, so these sub-flowsheets can inherit the fluid package of the

main flowsheet, or you can create an entirely new fluid package for each sub-flowsheet.

For each fluid package, you can define the following:

- Property package
- Components
- Reactions
- User properties

Figure 5.2



There is one common button at the bottom of the Simulation Basis Manager view.

Button	Description
Enter Simulation Environment	Enables you to enter the simulation environment of the UniSim Design case. Refer to Chapter 7 - Simulation Environment for more information.

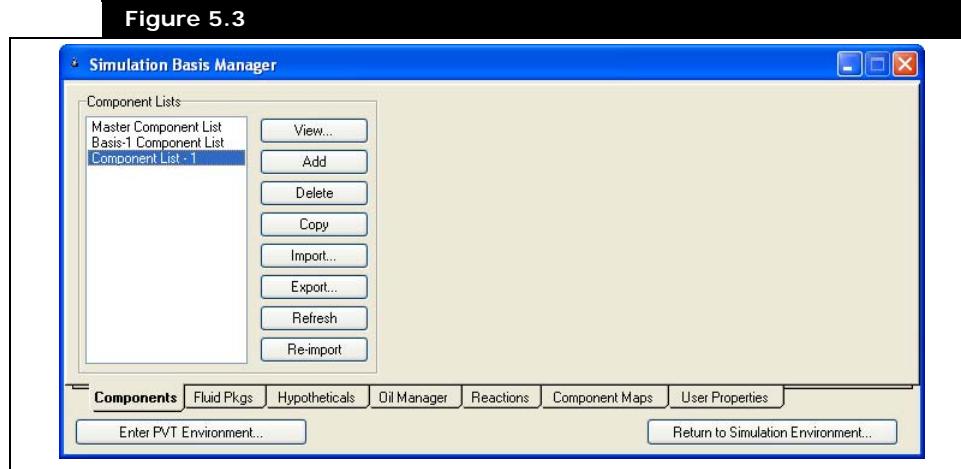
5.2.1 Components Tab

See [Chapter 1 - Components](#) in the **UniSim Design Simulation Basis Guide** for more information.

The Components tab is where you define the sets of chemical components used in the simulation. These component sets are stored in Component Lists and can include library pure components and/or

hypothetical components.

Figure 5.3



The Components tab contains a Master Component List that cannot be deleted. This master list contains every component available from "all" component lists. If you add components to any other component list, they are automatically added to the Master Component List. Also, if you delete a component from the master list, it is deleted from all other component lists using that component.

From this view, you can do the following to the component lists:

- View
- Add
- Delete
- Copy
- Import
- Export
- Refresh
- Reimport

Viewing a Component List

When you view a component list, you can add, remove, and sort the components in a list:

1. From the list of available component lists, select the component list you want to view/edit.
2. Click the **View** button. The selected Component List view appears.

Adding a Component List

To add a component list, click the **Add** button. The Component List view appears. This view allows you to add pure, electrolyte and hypothetical

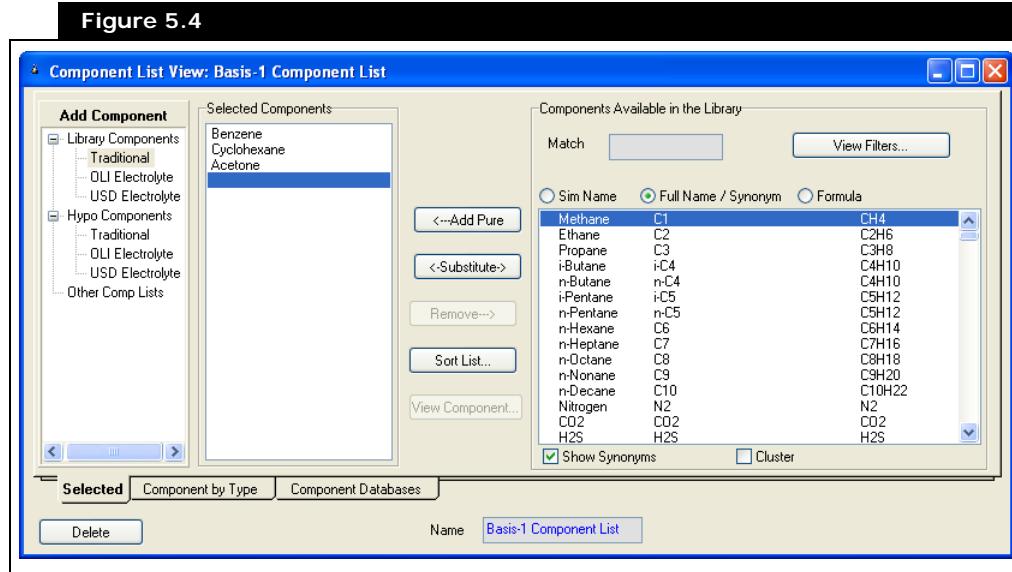
components to the new component list.

This view has three tabs: Selected, Component by Type and Component by Type.

Selected Tab

This tab allows you to add, remove, sort, and view components in a component list. This tab also provides a quick method for creating hypothetical groups and hypothetical components.

Figure 5.4



Adding a Traditional or Electrolyte Component

1. Click the **Selected** tab in the Component List view.
2. Double-click the Components branch in the Add Component tree to expand the tree. Two branches appear: Traditional and Electrolyte.
3. Click the Traditional or the Electrolyte sub-branch. The available components appear in the Component Library group.
4. In the **Match** field, type the name of the component you want to add to your list. UniSim Design filters the list of available components as you type.
5. In the component list, select the component you want to add.

The Electrolyte page won't show without "OLI Alliance Suite for UniSim" engine installed.

You can also double-click the component name to add it to the list of selected components.

6. Click the **Add Pure** button. The component is moved from the list of available components to the list of selected components.

Adding a Traditional or Electrolyte Hypothetical Component Group

1. Click the **Selected** tab in the Component List view.
2. In the Add Component tree, double-click the **Hypothetical** branch to expand the tree. Two sub-branches appear: Traditional and Electrolyte.
3. Click the Traditional or Electrolyte sub-branch. This displays the Hypothetical Components Available group. You have the option of adding either a group of hypothetical components or individual hypothetical components to your case.
4. To add a group of hypothetical components, select the Hypo Group you want to add from the list of available hypo groups.
5. Click the **Add Group** or **Add E Group** button available depending on whether you are in the Traditional or Electrolyte sub-branch.

Adding an Individual Hypothetical Component

6. From the Available Hypo Groups list, select the Hypo Group that contains the hypothetical component you want to add.
7. From the list of Available Hypo Components, select the hypothetical component you want to add.
8. Click the **Add Hypo** button in the Traditional sub-branch or **Add E Group** in the Electrolyte sub-branch.
 - The hypo component added into the Component List by clicking **Add E Hypo** or **Add E Group** will be renamed by replacing the * at the end of the hypo name with E and the Hypo ID number will be bumped up a value of 50000.
 - The **Quick Create a Hypo Component** button opens the Hypothetical Component Property view, which allows you to quickly create a new hypothetical component.
 - The **Quick Create a Solid Component** button opens the Hypothetical Solid Component Property view, which allows you to quickly create a new hypothetical solid component.

Refer to the [Section 5.4 - Component Property View](#) for more information about defining hypothetical components.

Adding Other Components

1. Click the **Selected** tab of the Component List view.
2. In the Add Component tree, select the Other branch. This displays the Existing Component Lists group.
3. From the list of available component lists, select the component list that contains the component you want to add.
4. From the list of available components, select the component you want to add to your case.
5. Click the **Add** button.

Removing Components

To remove a component from the component list:

1. Click the **Selected** tab of the Component List view.
2. Select the component you want to delete from the list of selected components.
3. Click the **Remove** button.

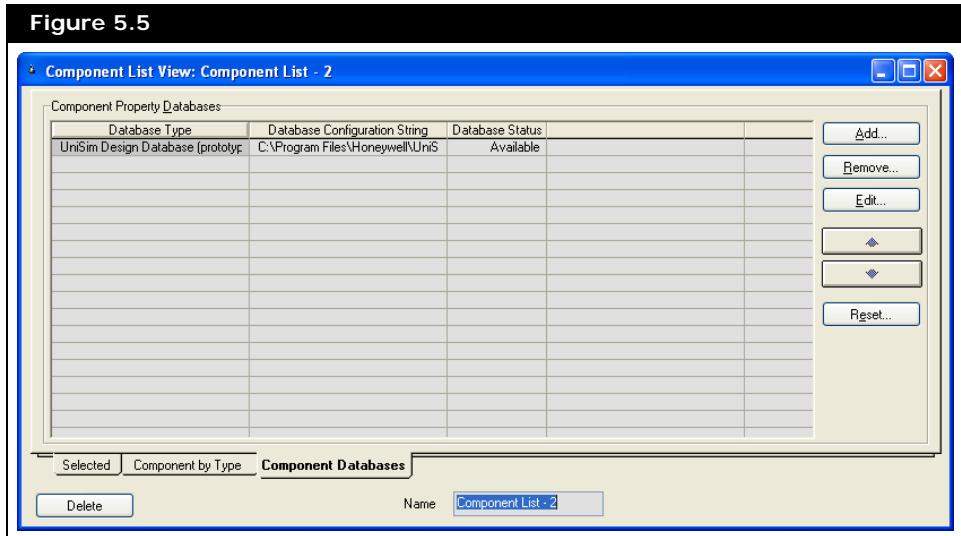
Component by Type Tab

This tab allows you to filter and view the components in your component list by type.

Component Databases

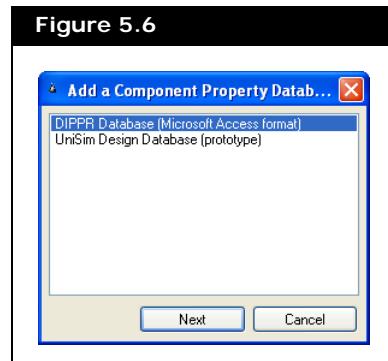
The Component Databases tab lets you to manage component property databases by adding, removing, resetting or ordering Component Property Databases. The component property database order will

decide the set of available components in the component list.



To Add a Component Property Database:

1. Click the **Add** button in the Component Property Databases group. The custom component property database type picker appears.



2. Choose the intended component property database type and Click **Next** to browse for Component Property Database file. The operation could be cancelled by clicking **Cancel**.
3. In the open file dialog, browse to the location of component property database file. From the list of available database files, select the file you want to use. Click **Open**. The selected component property database appears in the list of available component property database. You can also verify the database configuration string and database status of the added Component Property Database in Component Property Databases group.

To Remove a Component Property Database:

1. Select Component Property database to be removed from the

available database list.

2. Click the **Remove** button in the Component Property Databases group.
3. UniSim Design prompt will appear to confirm the removal of the selected Component Property Database. Click **Yes** to confirm or **No** to cancel the operation.

To Edit a Component Property Database:

1. Select Component Property database from the available Component Property database list to edit the properties.
2. Click the **Edit** button in the Component Property Databases group.
3. A File Open dialog box will appear to browse and select the edited or updated component property database file.

To Change the order of Component Property Database:

1. Select Component Property Database in Component Property Databases group for which sequence needs to be changed.
 2. Click the Up Arrow button to move the selected Component Property database higher in the order of the list.
 3. Click the Down Arrow button to move the selected Component Property database lower in the order of the list.
1. To **Reset** Component Property Databases:
 2. Click the Reset button in the Component Property Databases group.
 3. A dialog box will prompt you to confirm reset for the list of Component Property Databases. Click **Yes** to confirm or **No** to cancel the operation.

Changing the order of database loader is possible only when more than one Component Property Database is configured.

Component property databases will be reset as per the UniSim Design Session preferences.

The Master Component list cannot be deleted.

Deleting a Component List

1. From the list of available component lists, select the component list you want to delete.
2. Click the **Delete** button.

UniSim Design does not prompt you to confirm the deletion of your component list. After the list is deleted, the information cannot be retrieved.

Copying a Component List

This procedure assumes you are in the main Simulation Basis Manager view.

1. From the Component List, select the name of the component list you want to copy.
2. Click the **Copy** button.

Copying a component list creates a new component list with the exact same properties as the original.

Importing a Component List

1. On the Simulation Basis Manager view, click the **Import** button. The Open File view appears.
2. Browse to the location of your component list file (*.cml).
3. Select the file you want to import, then click **Open**.

Exporting a Component List

1. On the Simulation Basis Manager view, click the **Export** button. The Save File view appears.
2. Specify the name of your component list file and its location.
3. Click **Save**.

Refreshing a Component List

1. On the Simulation Basis Manager view, click the **Refresh** button.
2. If any library component in the master component list has any changed property values, a dialog box will be displayed to ask your permission to overwrite the values with library values for that component.

5.2.2 Fluid Packages Tab

See [Chapter 2 - Fluid Package](#) in the [UniSim Design Simulation Basis](#) guide for more information.

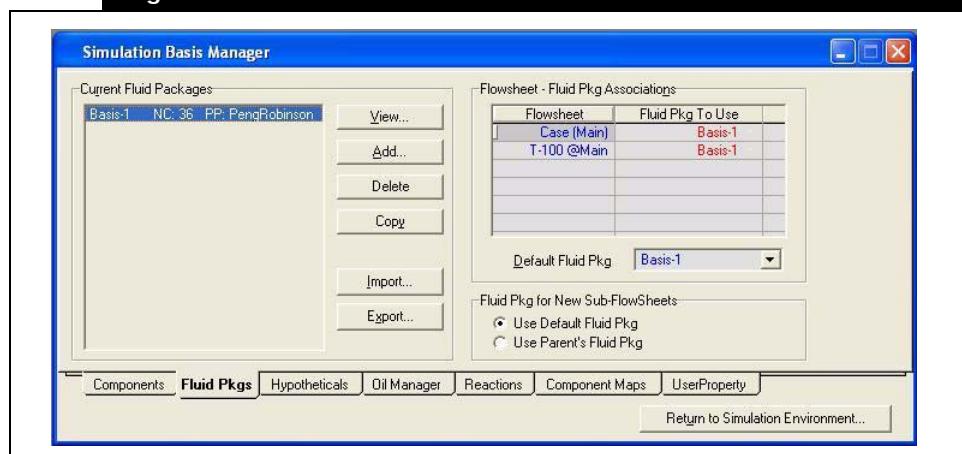
The fluid package contains all the necessary information for pure component flash and physical property calculations. This allows you to define all the required information inside a single entity. There are four key advantages to using fluid packages:

- All associated information is defined in a single location for easy creation and modification.
- Fluid packages can be exported and imported as completely defined packages for use in any simulation.

- Fluid packages can be cloned, reducing the time involved in creating and/or modifying complex fluid packages.
- Multiple fluid packages can be used in the same simulation.

The Fluid Pkgs tab of the Simulation Basis Manager allows you to create and manipulate multiple fluid packages.

Figure 5.7



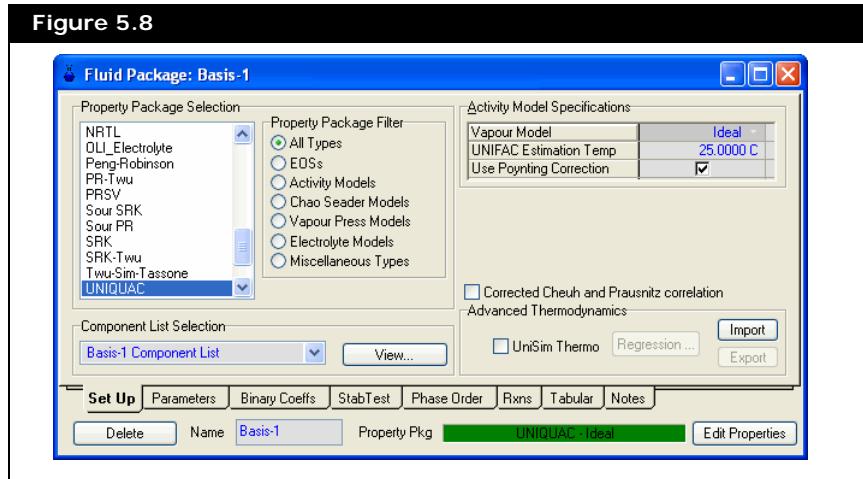
Each fluid package available to your simulation appears in the Current Fluid Packages group with the following information displayed:

- Name
- Number of components attached to the fluid package
- Property package attached to the fluid package

Adding a Fluid Package

1. In the Simulation Basis Manager view, click the **Add** button. The

Fluid Package screen appears as shown below.



2. Click the **Set Up** tab.
3. From the Component List Selection drop-down list, select the component list you want to use for your fluid package.
4. From the list of available property packages, click the property package you want to use. If the property package is one of the following, then additional property package options appear in the top right corner of the view:
 - Equation of State (EOS)
 - Activity Model
 - Amines package
 - Electrolyte property package

The rest of the tabs in the Fluid Package Manager view are used to modify the fluid package according to your requirements.

Editing a Fluid Package

1. From the Current Fluid Packages list, select the fluid package that you want to edit.
2. Click the **View** button to display the Fluid Package view.
3. Modify any of the parameters that comprise the fluid package.

Deleting a Fluid Package

1. From the Current Fluid Packages list, select the fluid package that

you want to delete.

2. Click the **Delete** button.

UniSim Design does not prompt you to delete the last fluid package from the Current Fluid Packages list.

Copying a Fluid Package

1. From the Current Fluid Packages list, select the fluid package you want to copy.
2. Click the **Copy** button.

Copying a fluid package creates a new fluid package with the exact same properties as the original.

Importing a Fluid Package

1. In the Current Fluid Packages group, click the **Import** button. The Open File view appears.
2. Browse to the location of your fluid package file (*.fpk).
3. Select the file you want to import and click **Open**.

Exporting a Fluid Package

1. In the Current Fluid Packages group, click the **Export** button. The Save File view appears.
2. Specify the name and location of the fluid package file.
3. Click **Save**.

Associating a Fluid Package with a Flowsheet

1. In the Flowsheet-Fluid Pkg Associations group, select the required flowsheet.
2. Click the corresponding **Fluid Pkg To Use** cell to open a drop-down list.
3. From the list, click the fluid package you want to associate with that flowsheet.

There must be at least one fluid package within a case before you can assign it to a flowsheet. The Default Fluid Pkg drop-down list specifies which fluid package to use as the default. The default fluid package is automatically assigned to any new flowsheets that are created within the case.

5.2.3 Hypotheticals Tab

See [Chapter 4 - Hypotheticals](#) in the UniSim Design Simulation Basis Guide for more information.

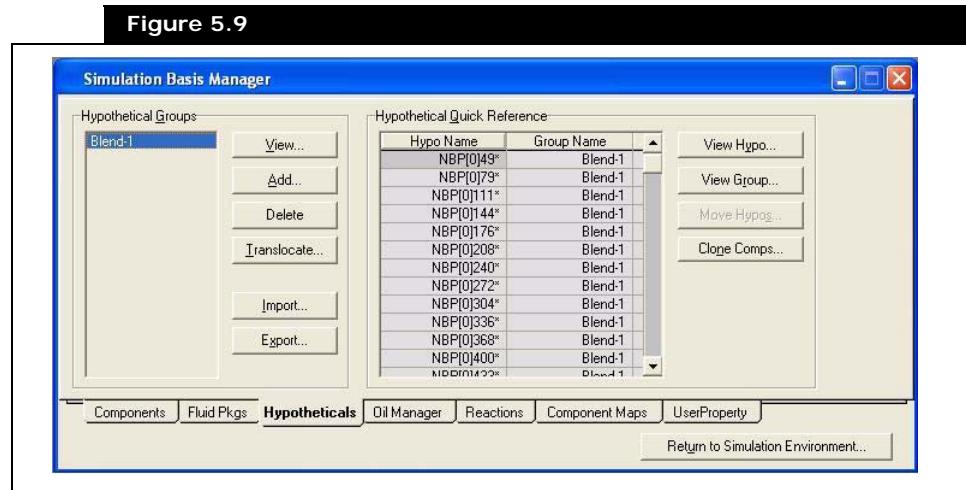
This tab enables you to create non-library or hypothetical components. Hypothetical components can be any of the following:

- Pure components
- Defined mixtures
- Undefined mixtures
- Solids

You can also clone library components into hypothetical components, which allows you to modify the library values.

When you click the Hypotheticals tab, the following view appears.

Figure 5.9



A broad selection of estimation methods are provided for the various Hypo groups, ensuring the best representation of the behaviour of the hypothetical components in the simulation. In addition, methods are provided for estimating the interaction binaries between hypotheticals and library components. You can also use hypotheticals with the Tabular Package and in reactions.

Hypothetical components are independent of the fluid package, and when created, they are placed in a Hypo Group. Since hypothetical components are not exclusive to a particular fluid package, multiple fluid packages can share hypotheticals. You can create a hypothetical component or group once and use it in any fluid package within the case.

Adding a Hypothetical Group

1. Click the **Add** button. The Hypo Group view appears.
2. From the Component Class drop-down list, select the class for grouping your hypotheticals.
3. Click either the **Add Hypo** or **Add Solid** button. Keep clicking the button until all of the hypothetical components are added to the group.
4. Enter the information from the following table for each component so UniSim Design can estimate the properties of the components.

Normal Boiling Point	Minimum Required Information
< 700°F (370°C)	Normal Boiling Point
> 700°F (370°C)	Normal Boiling Point and Liquid Density
Unknown	Molecular Weight and Liquid Density

5. Each component also requires a UNIFAC structure. Click the **UNIFAC** button to display the UNIFAC Component Builder view.

Deleting a Hypothetical Component

1. From the list of available hypothetical components, select the component you want to delete.
2. Click the **Delete** button.

UniSim Design does not prompt you to confirm the deletion of your hypothetical component. After the component is deleted, the information cannot be retrieved.

Cloning Library Components

Use this procedure to clone a library component into a hypothetical component.

Select more than one component by holding down the **CTRL** key and clicking each component you want to select.

1. In the Hypo Group view, click the **Clone Library Comps** button. The Convert Library Comps to Hypothetical Comps view appears.
2. The Source Components group has two lists. From the list of available component lists, select the component list that contains the component you want to clone.
3. From the lists of available library components, select the component you want to clone.
4. In the Hypo Groups group, select the target hypo group for your new hypothetical component.
5. Click the **Convert to Hypo(s)** button. This clones the selected component(s).

Editing a Hypothetical Group

1. From the list of available hypothetical groups, select the hypothetical group you want to edit.
2. Click the **View** button to display the Hypo Group view.
3. From here you can add and remove hypothetical components, change the component class, clone library components and change estimation methods.

You can also access the Hypo Group view by selecting a hypothetical component from the Hypothetical Quick Reference table and clicking the **View Hypo** button. The information that appears corresponds to the hypo group the hypothetical component is associated with.

Deleting a Hypothetical Group

1. From the list of available hypothetical groups, select the hypothetical group you want to delete.
2. Click the **Delete** button.

Importing a Hypothetical Group

1. Click the **Import** button. The Open File view appears.
2. Browse to the location of your hypothetical group file (*.hyp).
3. Select the file you want to import, then click the **Open** button.

Exporting a Hypothetical Group

1. Click the **Export** button. The Save File view appears.
2. Specify a name for your hypothetical group file and the location of your file.
3. Click **Save**.

5.2.4 Oil Manager Tab

See [Chapter 3 - UniSim Design Oil Manager](#) in the [UniSim Design Simulation Basis Guide](#) for more information.

The Oil Characterization environment is where the characteristics of a petroleum fluid can be represented by using discrete hypothetical components. Physical, critical, thermodynamic and transport properties are determined for each hypothetical component using correlations that you select. The fully defined hypocomponent can then be installed in a stream and used in any flowsheet.

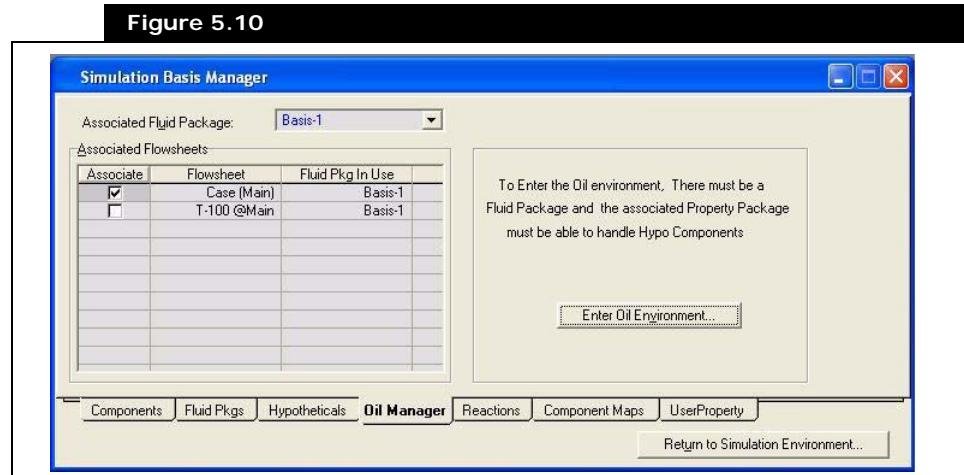
To use the Oil Characterization environment, at least one fluid package must exist in the case. Any hypothetical components must be

compatible with the property method used by the fluid package.

UniSim Design defines the hypocomponent by using assay data you provide. The following are features exclusive to the oil environment:

- Providing laboratory assay data
- Cutting a single assay
- Blending multiple assays
- Assigning a user property to hypocomponents
- Selecting correlation sets to determine properties
- Installing hypocomponents into a stream
- Viewing tables and plots for your input and for the characterized fluid

Figure 5.10



5.2.5 Reactions Tab

See [Chapter 5 - Reactions](#) in the [UniSim Design Simulation Basis Guide](#) for more information.

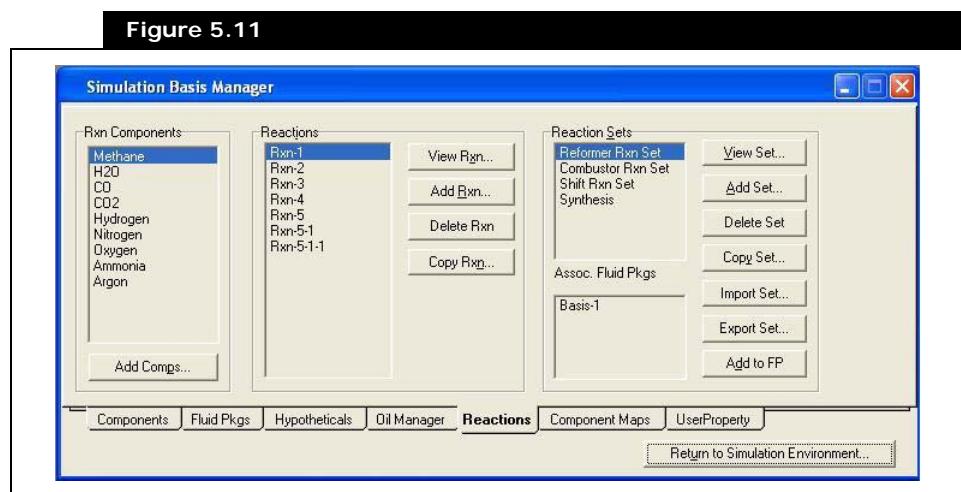
The Reactions Tab in the Simulation Basis Manager allows you to define reactions within UniSim Design. You can define an unlimited number of reactions and group these reactions in reaction sets. The reaction sets are then attached to unit operations in the flowsheet.

Any ReactionSet and Reaction in the Reaction Manager bank cannot be attached to any unit operation in an electrolyte flowsheet (reactor unit operations are disabled).

The electrolytes thermo calculation conducts a reactive and phase flash at the same time. Therefore, adding any external reactions to a unit operation is not yet allowed in UniSim Design for electrolyte simulation.

For more information, refer to the [UniSim Design OLI Interface Reference Guide](#).

The Reaction tab appears as shown in the following figure.



Use the Reaction Manager to do the following:

- Create a new list of components for the reactions or use the components associated with a fluid package.
- Add, Edit, Copy or Delete reactions and reaction sets.
- Attach reactions to various reaction sets, or attach reaction sets to multiple fluid packages.
- Import and Export reaction sets.

Adding a Reaction

1. Click the **Add Rxn** button. The Reactions view appears.
2. Select the type of reaction that you want to use.
3. Click the **Add Reaction** button. The Reaction Property view appears; in this view, you can define the following:
 - Stoichiometry
 - Conversion basis
 - Equilibrium constant
 - Other properties
4. Click the **Stoichiometry** tab.
5. Click the field that displays ****Add Comp****. Select the component you want to use for the reaction from the drop-down list.
6. Repeat the previous step until all of the required components are added to the table.

7. In the Stoich Coeff column, enter a stoichiometric coefficient for each component. This value must be negative for a reactant and positive for a product.
8. Specify the coefficient for an inert component as 0 (which for the Conversion reaction is the same as not including the component in the table). Fractional coefficients are acceptable.

Editing a Reaction

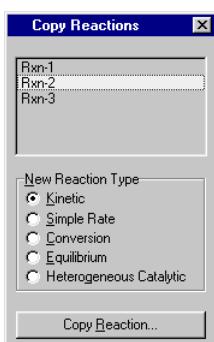
1. From the list of available reactions, select the reaction you want to edit.
2. Click the **View Rxn** button. The Reaction Property view appears. In this view, you can modify the following:
 - Stoichiometry
 - Conversion basis
 - Equilibrium constant
 - Other properties

Deleting a Reaction

1. From the list of available reactions, select the reaction you want to delete.
2. Click the **Delete Rxn** button. UniSim Design prompts you to confirm the deletion.

Copying a Reaction

1. From the list of available reactions, select the reaction you want to copy.
2. Click the **Copy Rxn** button. The Copy Reactions view appears.
3. Select the reaction you want to copy from the list of reactions.
4. Use the radio buttons in the New Reaction Type group to select the reaction type for the reaction copy.
5. Click the **Copy Reaction** button.



Adding a Reaction Set

1. Click the **Add Set** button. The Reaction Set view appears.
2. In the Active List column, click the <empty> cell and use the drop-down list to select the reaction you want to add to the set.
3. In the Inactive List column, click the <empty> cell and use the drop-down list to select the reaction you want to add to the set. This reaction remains inactive, but it is included in the set.

Available reaction solver methods for Kinetic type reaction set:

- Newton's Method
- Rate Iterated
- Rate Integrated
- Auto Select

4. For a Kinetic type reaction set, you may select a solver method. From the Solver Method drop-down list, select the reaction solver method you want to use.
5. Add any of the available reactions to the set (as long as they are the same type). A single reaction can be added to as many sets as necessary.

Editing a Reaction Set

1. From the list of available reaction sets, select the reaction set you want to edit.
2. Click the **View Set** button. The Reaction Set view appears. In this view, you can do the following:
 - Add and remove reactions in the reaction set.
 - Modify the solver method.
 - Activate and deactivate reactions already in the set.

Deleting a Reaction Set

1. From the list of available reaction sets, select the reaction set you want to delete.
2. Click the **Delete** button. UniSim Design prompts you to confirm the deletion of the reaction set.

Copying a Reaction Set

1. From the list of available reaction sets, select the reaction set you want to copy.
2. Click the **Copy** button.

Copying a reaction set creates a new reaction set with the exact same properties as the original.

Importing a Reaction Set

1. Click the **Import Set** button. The Open File view appears.
2. Browse to the location of your reaction sets file (*.rst).
3. Select the file you want to import, then click **Open**.

Exporting a Reaction Set

1. Click the **Export Set** button. The Save File view appears.
2. Specify the name and location of your reaction set file.
3. Click **Save**.

Adding a Reaction Set to a Fluid Package

After creating reactions and reaction sets, you can associate the set(s) with a fluid package.

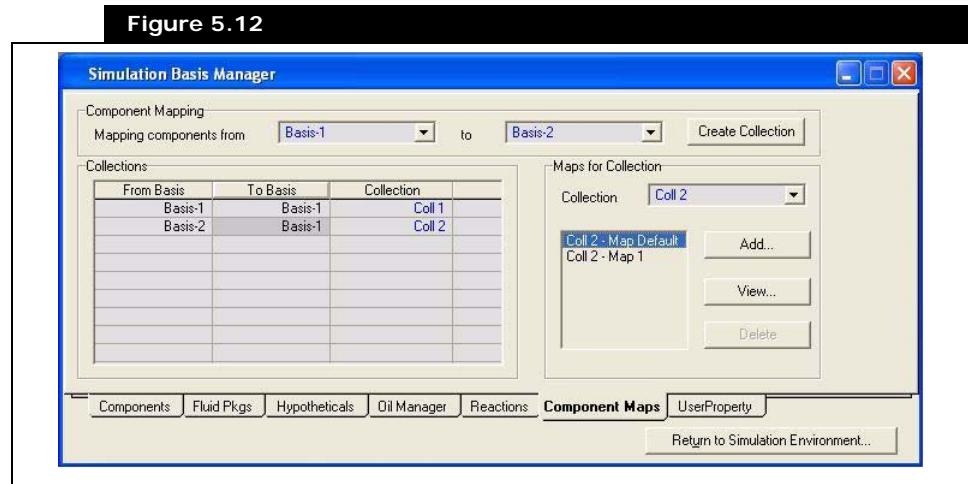
1. Click the **Add to FP** button. The Add Reaction Set view appears.
2. From the list of available fluid packages, select the fluid package to which you want to add a reaction set.
3. Click the **Add Set to Fluid Package** button.

5.2.6 Component Maps Tab

See [Chapter 6 - Component Maps](#) in the **UniSim Design Simulation Basis Guide** for additional information.

The Component Maps tab allows you to 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 useful when dealing with hypothetical oil components.

Figure 5.12



Two previously defined fluid packages are required to perform a component mapping. One fluid package becomes the target component set and the other becomes the source component set. Mapping is performed using a matrix of source and target components. The transfer basis can be performed on a mole, mass or liquid volume basis.

Adding a Collection

1. In the Component Mapping group, select the fluid packages you want to map.
2. After two distinct fluid packages are selected, click the **Create Collection** button to add a new collection to the Collections table.

The table lists the following information:

- The fluid package where the components came from
- The fluid package where the components are going
- The collection name (can be edited at any time)

Adding a Component Map

1. From the Collection drop-down list in the Maps for Collection group, select the collection for which you want to map components.
2. Click the **Add** button. The Component Map view appears.
3. In the component matrix, map all specifiable components (values in red text). All source components appear in columns and all target components appear in rows.
4. In the Transfer Basis group, select the basis of the component transfer by selecting the **Mole**, **Mass**, or **Volume** radio button.

Editing a Component Map

1. From the Collection drop-down list in the Maps for Collection group, select the required collection.
2. From the list of available component maps, select the component map you want to edit.
3. Click the **Edit** button. The Component Map view appears. In this view, you can modify the parameters for the map.

Deleting a Component Map

1. From the Collection drop-down list in the Maps for Collection group, select the collection for which you want to delete component map.
2. From the list of available component maps, select the component map you want to delete.
3. Click the **Delete** button.

You are not prompted to confirm the deletion, so ensure you have selected the correct component map before deleting.

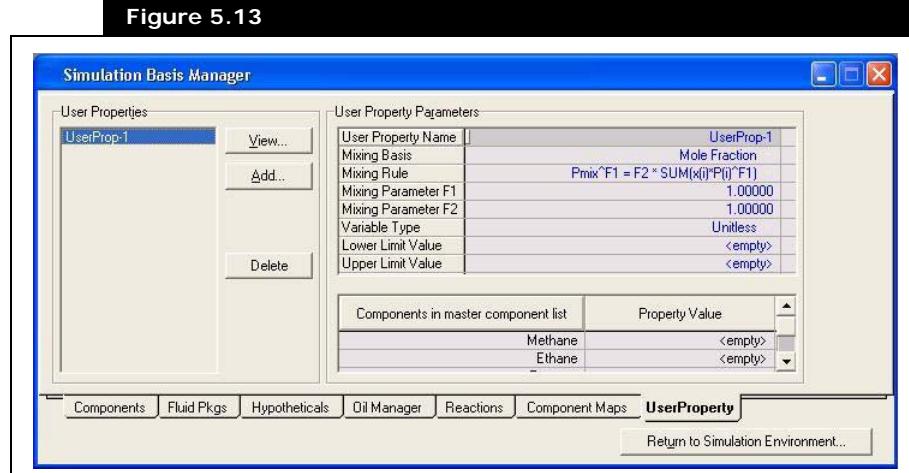
5.2.7 User Property Tab

See [Chapter 7 - User Properties](#) in the [UniSim Design Simulation Basis Guide](#) for more information.

Use the User Property tab to create user properties for use in the

Simulation environment.

Figure 5.13



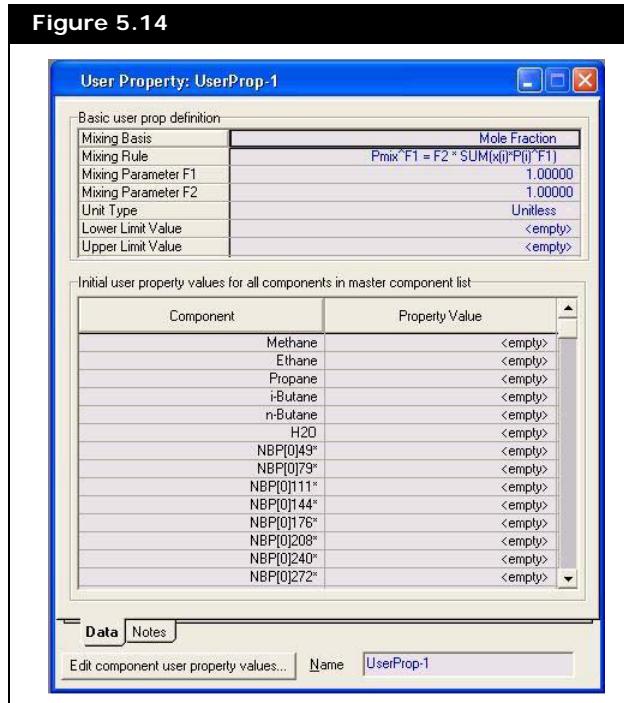
You can also create user properties in the Oil Characterization environment. See [Section 6.2.4 - User Property Tab](#) for more information.

User properties are any property that can be defined and calculated on the basis of composition. You supply a user property value for each component in a fluid package, then select the mixing basis and mixing equation to calculate the total user property.

When a user property is defined, UniSim Design calculates the value of the property for any stream through the user property utility. User properties can also be set as Column specifications.

Adding a User Property

1. Click the **Add** button. The User Property view appears.
2. Click the **Data** tab.



3. In the **Mixing Basis** field, use the drop-down list *i* to select the basis for mixing.
4. In the **Mixing Rule** field, use the drop-down list to specify the mixing rule to use with for your user property.
5. Specify values for the mixing rule parameters F1 and F2 to accurately reflect your property formula. By default these parameters have a value of 1.00.
6. Select a unit from the Unit Type drop-down list for the user property value. If the unit type is Temperature, the internal unit used is degrees Kelvin. (This is important when calculating the mixed value for a stream.)
7. Specify a lower and upper limit for your user property in the Lower Limit Value and Upper Limit Value cells.

Refer to the **UniSim Design Simulation Basis Guide** for more information regarding the mixing rules.

The property parameters determine how the user property is calculated in all streams. Whenever the value of a user property is requested by the User Property utility or by the Column, UniSim Design uses the composition in the specified basis and calculates the user property value using the mixing rules and parameters that you set.

Editing a User Property

- From the list of available user properties, select the user property that you want to edit.
- Click the **View** button. The User Property view appears.

In this view, you can edit all of the parameters that define the user property.

Deleting a User Property

- From the list of available user properties, select the user property that you want to delete.
- Click the **Delete** button.

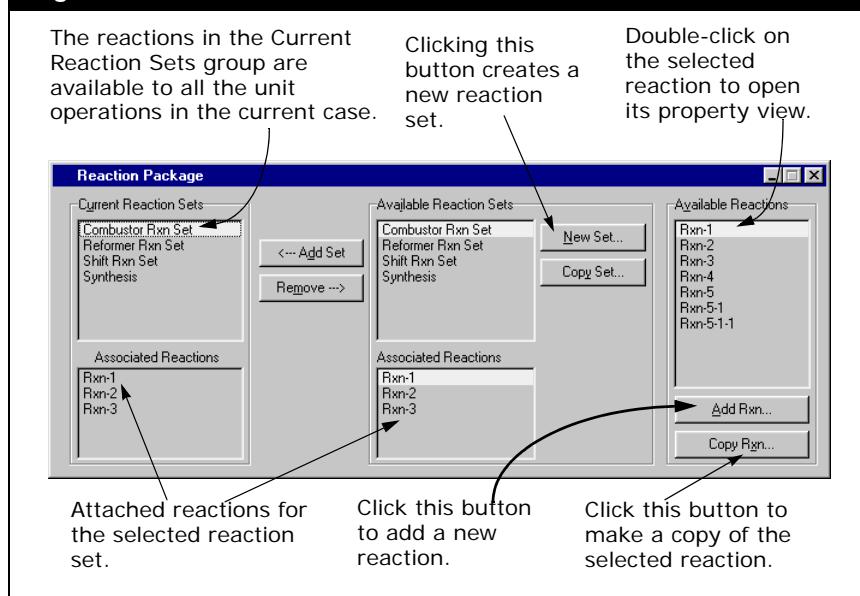
You are not prompted to confirm the deletion of your user property, so ensure you are deleting the correct one. After a user property is deleted, it cannot be retrieved.

5.3 Reaction Package

Refer to [Section 5.2.5 - Reactions Tab](#) and [Chapter 5 - Reactions](#) in the **UniSim Design Simulation Basis Guide** for more information about reactions and reaction sets.

From the Basis menu, select Reaction Package to open the Reaction Package view. Use this view to create, copy and edit both reactions and reaction sets. You can attach reactions to a reaction set and make a Reaction Set available to unit operations within the current case.

Figure 5.15



The Reaction Package view eliminates the need to return to the Reaction tab of the Simulation Basis Manager when defining reactions and reaction sets. The only task when defining reactions that must be done in the Simulation Basis Manager is the selection of components.

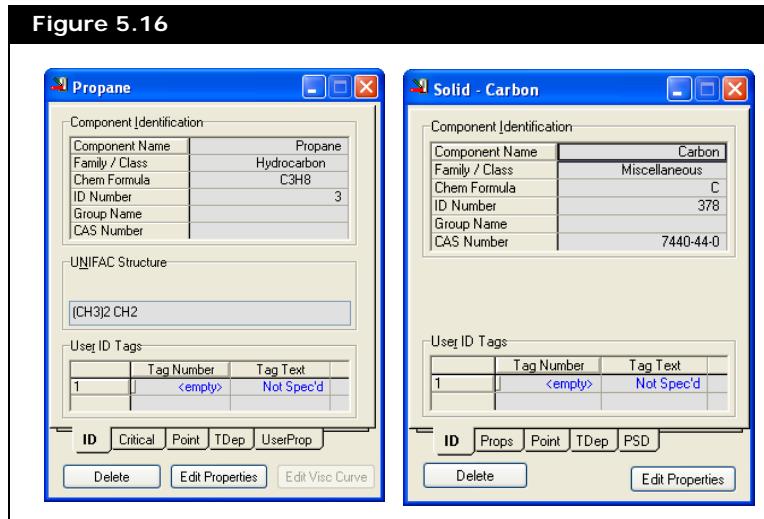
5.4 Component Property View

The Component Property view enables you to view and edit the properties of a component. The two types of property views are the traditional component property view and solid component property view.

Refer to [Chapter 1 - Components](#) in the **UniSim Design Simulation Basis Guide** for more information.

You can access both types of property views from the Component List view. You can also access the property view for hypothetical and solid hypothetical components from the Hypo Group view.

Refer to the following figure to see how both the Traditional and Solid pure component views appear.



Both of these views contain five tabs and the information is colour coded according to the type of information being displayed. The following table explains the color coding.

Colour	Information description
Red	Values estimated by UniSim Design.
Blue	User defined values.
Black	Calculated values or information that you cannot modify.

You can supply values for any of the component properties, or overwrite

values estimated by UniSim Design. If you change a specified value, all properties previously estimated using that specification are removed.

5.4.1 Viewing a Pure Component

Use this procedure to view the properties of a pure component found in a component list.

1. Select the pure component you want to view from the list of available components.
2. Click the **View Component** button. The Component Property view appears for the selected component.

Accessing a Hypothetical Component Property View

You can access the hypothetical component property view using the following methods:

- In the Component List view, select the component from the list of selected components and click the **View Component** button.
- On the Hypotheticals tab of the Simulation Basis Manager, select the component from the Hypothetical Quick Reference table and click the **View Hypo** button.
- In the Hypo Group view, select the component from the list of available components and click the **View** button.

5.4.2 Defining a Traditional Hypothetical Component

Refer to the sections [Adding a Traditional or Electrolyte Hypothetical Component Group](#) and [Adding a Hypothetical Group](#) for more information.

1. Open the component property view, or create a new hypothetical component.
2. Click the **ID** tab.
3. Click the **Structure Builder** button. The UNIFAC Component Builder view appears.
4. From the list of available UNIFAC groups, select the sub-group you want to add to your structure.
5. Click the **Add Groups** button.
6. Repeat steps 4 and 5 until you have the structure you want and there are no free bonds.
7. Close the UNIFAC Component Builder view.
8. Click the **Critical** tab.

9. Enter the information from the following table so UniSim Design can estimate the properties of your component.

Normal Boiling Point	Minimum Required Information
< 700°F (370°C)	Normal Boiling Point
> 700°F (370°C)	Normal Boiling Point and Liquid Density
Unknown	Molecular Weight and Liquid Density

10. After entering the values, click the **Estimate Unknown Props** button to calculate the other properties.

5.4.3 Defining a Solid Hypothetical Component

Refer to the sections [Adding a Traditional or Electrolyte Component Group](#) and [Adding a Hypothetical Group](#) for more

1. Open the component property view, or create a new hypothetical component.
2. Click the **Critical** tab.
3. Enter both the molecular weight and density (at minimum).
4. Click the **Estimate Unknown Props** button to calculate the other properties.

6 Oil Characterization Environment

6.1 Introduction	2
6.2 Oil Characterization Manager.....	2
6.2.1 Oil Output Settings View	3
6.2.2 Assay Tab	4
6.2.3 Cut/Blend Tab.....	6
6.2.4 User Property Tab	9
6.2.5 Correlation Tab	10
6.2.6 Install Oil Tab	13

6.1 Introduction

Refer to [Chapter 3 - UniSim Design Oil Manager](#) in the [UniSim Design Simulation Basis Guide](#) for additional information.

The Oil Characterization environment enables you to characterize petroleum fluids by creating and defining Assays and Blends. The oil characterization procedure generates petroleum hypocomponents for use in your fluid package(s). The Oil Characterization environment is accessible only through the Simulation Basis environment.

To enter the Oil Characterization environment, at least one fluid package must exist in the case and any hypothetical components must be compatible with the property method used by the fluid package.

The Desktop for the Oil Characterization environment is similar to the Desktop in the Simulation Basis environment. Icons specific to generating oils appear and the Oil Characterization Manager is the Home View.

Accessing the Oil Characterization Environment



Oil Environment icon

To enter the Oil Characterization environment, click the Oil Environment icon or the Enter Oil Environment button found in the Oil Manager tab of the Simulation Basis Manager.

6.2 Oil Characterization Manager

The Oil Characterization Manager contains five tabs and four buttons. The buttons are as follows:

- **Clear All.** Click to clear all the information entered or imported in the Oil environment.
- **Calculate All.** Click to calculate all unknown variables. This button is only effective if you supply enough information about the hypocomponents.
- **Oil Output Settings.** Click to open the Oil Output Settings view.
- **Return to Basis Environment.** Click to return to Simulation Basis environment.

Refer to [Section 1.10 - Petroleum Fraction](#) in the [UniSim Design OLI Interface Reference Guide](#) for more information about the OLI_Electrolyte Fluid Package.

6.2.1 Oil Output Settings View

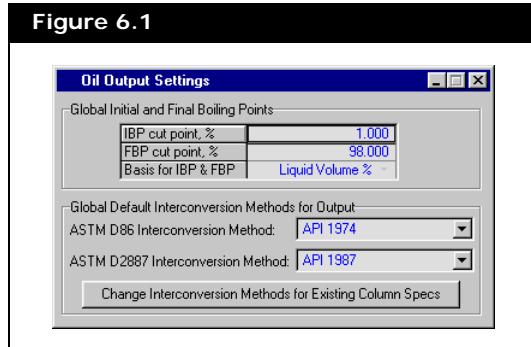
To access this view, click the Oil Output Settings button on the Oil Characterization view.

Refer to [Chapter 14 - Utilities](#) in the **UniSim Design Operations Guide** for more information about the BP Curves utility.

During the characterization of an oil or the calculation of boiling ranges for a fluid in the BP Curves utility, the initial boiling point (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's boiling point regions, the most difficult to measure are the end regions.

UniSim Design uses the defaults of 1% and 98% on liquid volume basis for the IBP and FBP. With a 1% IBP value, UniSim Design uses the boiling points of all components in the first volume percent of the given fluid and calculates a weighted average boiling point that is used as the IBP for any further analysis.

The final boiling point is determined similarly, using the weighted average of the boiling points for the components found in the final two liquid volume percents of the fluid.



These IBP and FBP values can be modified in this view. In the IBP cut point and FBP cut point fields, enter the values you want to use for calculating your BP curve.

Refer to the following table for a description of the values.

Field	Description
IBP cut point	Specify a value from 0% to 5%.
FBP cut point	Specify a value from 90% to 100%.
Basis for IBP & FBP	Using the drop-down list, select the basis for your boiling points. You have three choices: Liquid Volume%, Mass%, and Mole%.

You can also specify the Interconversion method used for the ASTM

D86 interconversion and ASTM D2887 interconversion. Select the method you want to use.

For the ASTM D86 Interconversion Method, the following options are available:

- API 1974
- API 1987
- API 1994
- Edmister-Okamoto 1959

For the ASTM D2887 Interconversion Method, the following options are available:

- API 1987
- API 1994 Indirect
- API 1994 Direct

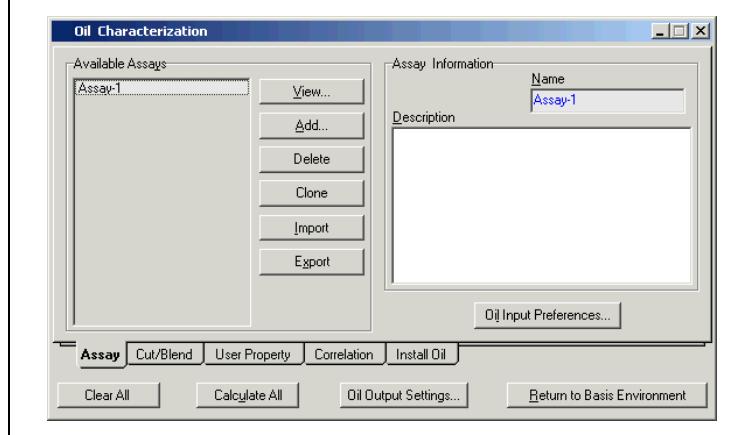
Note: OLI_Electrolyte property package does not support Oil Output Settings. For details, refer to [Section 1.10 - Petroleum Fraction](#) in the UniSim Design OLI Interface Reference Guide

6.2.2 Assay Tab

See [Section 3.5 - Characterizing Assays](#) in the [UniSim Design Simulation Basis Guide](#) for additional information.

The Assay tab allows you to manage the oil assays in your case. Use this tab to add new assays and modify existing ones. You can also import assays from other cases or export them for use with another case. The Description field lets you add notes for individual assays.

Figure 6.2



The minimum amount of information required to characterize a petroleum fluid is either a laboratory distillation curve or two of the following three bulk properties:

- Molecular Weight
- Density
- Watson UOP K factor

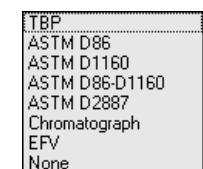
The more information you supply, the better the results will be for your oil characterization.

All physical and critical properties are generated from an internally generated TBP curve at atmospheric conditions. Regardless of the assay data provided, UniSim Design converts it to an internal TBP curve for the characterization procedure. This internal TBP curve is not stored with the assay.

Click the Oil Input Preferences button to change the default oil preferences. Refer to [Section 12.8 - Oil Input Tab](#) for more information.

Adding an Assay

1. In the Oil Characterization view, click the **Add** button. The Assay Property view appears.
2. Click the **Input Data** tab.
3. In the Assay Definition group, use the Bulk Properties drop-down list to specify if you are supplying bulk properties. Select either **Not Used** or **Used**.
4. From the Assay Data type drop-down list, select an assay data type option.
5. Depending on the assay data type selected, supply information for the following:
 - Light Ends curve
 - Molecular Weight curve
 - Density curve
 - Viscosity curves
 - Distillation conditions
6. Each definition requires you to provide data in the Input Data group. Click the required radio button and either enter the data directly into the table or click the **Edit Assay** radio button.
7. Click the **Light Ends Handling & Bulk Fitting Options** button to specify if a given curve contains light-ends contributions. Also, specify whether the specified bulk properties contains light-ends and partition a property curve, so that some sections can be adjusted more than others.



Assay Data Type options

Editing an Assay

1. From the list of available assays, select the assay you want to edit.
2. Click the **View** button. The Assay Property view appears.
3. Click the **Input Data** tab, then modify the assay data.
4. Click the **Calculation Defaults** tab, then modify how the hypocomponents are calculated.

Deleting an Assay

1. From the list of available assays, click the assay you want to delete.
2. Click the **Delete** button.

You will not be prompted to confirm the deletion of an assay, however, assays being used by a blend will not be deleted.

Cloning an Assay

1. From the list of available assays, select the assay you want to clone.
2. Click the **Clone** button. Cloning creates a new assay with the exact same properties as the original.

Importing an Assay

1. Click the **Import** button. The Open File view appears.
2. Browse to the location of your assay file (*.oil).
3. Select the file you want to import and click **Open**. The new assay appears in the list of available assays.

Exporting an Assay

1. Click the **Export** button. The Save File view appears.
2. Specify a name and location for your assay file.
3. Click **Save**.

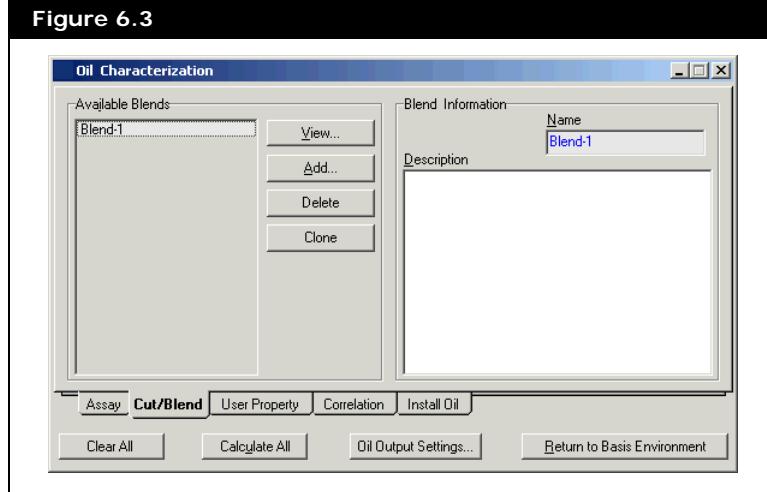
6.2.3 Cut/Blend Tab

See [Section 3.6 - Hypocomponent Generation](#) in the [UniSim Design Simulation Basis Guide](#) for additional information.

The Cut/Blend tab allows you to manage the hypocomponents in a

case.

Figure 6.3



The Cut/Blend tab allows you to do the following:

- Add new blends
- Modify existing blends
- Clone existing blends
- Provide descriptions for individual blends so others can easily access the information.

The Cut/Blend characterization splits internal working curves for one or more assays into hypocomponents.

To modify the graph labels in a Blend plot, click the **Clone** button. UniSim Design generates a copy of the existing selected plot. The cloned plot is independent, and its labels can be modified and are not overwritten.

For more information, refer to [Chapter 3 - UniSim Design Oil Manager](#) in the [UniSim Design Simulation Basis Guide](#).

Adding a Blend

1. Click the **Add** button. The Blend Property view appears.
2. Click the **Data** tab.
3. From the list of available assays, select the assay you want use for the blend.
4. Click the **Add** button. The assay is moved from the list of available assays to the Oil Flow Information table.
5. In the Oil Flow table, specify the flow units for the oil as either Liquid Volume, Molar, or Mass.
6. Specify the flow rate of the oil.

7. From the Cut Option Selection drop-down list, select one of the following cut options:
 - Auto Cut
 - User Ranges
 - User Points

There is no limit for the number of assays that can be included in a single blend or the number of blends that can contain a given assay. Each blend is treated as a single oil and does not share hypocomponents with other blends or oils.

The Bulk Data button is available when more than one assay is present in the Oil Flow Information table. Clicking this button opens the Bulk Values view, where you can provide the following bulk data for a blend:

- Molecular Weight
- Mass Density
- Watson (UOP) K
- Viscosities at two temperatures.

The Bulk Data feature is useful for supplying the bulk viscosities of the blend if they are known.

Editing a Blend

1. From the list of available blends, select the blend you want to edit.
2. Click the **View** button. The Blend Property view appears.
3. Click the **Data** tab, then modify the data for the blend.

Deleting a Blend

1. From the list of available blends, click the blend you want to delete.
2. Click the **Delete** button.

UniSim Design will not prompt you to confirm the deletion of a blend, so ensure you have selected the correct blend before deleting.

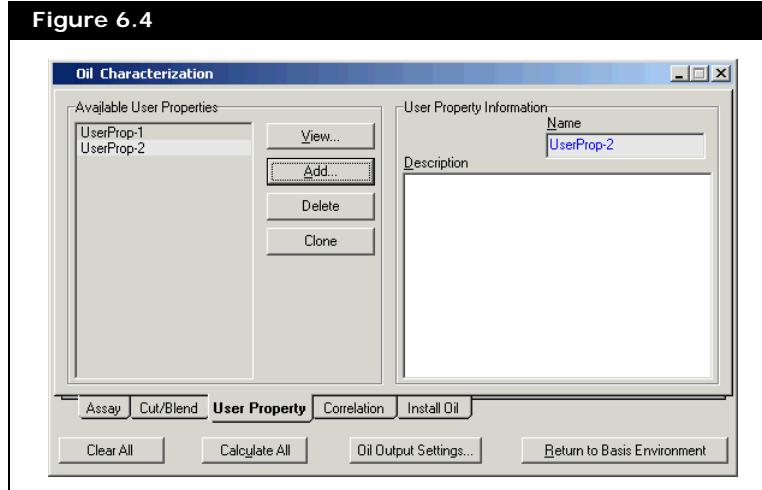
Cloning a Blend

1. From the list of available blends, select the blend you want to clone.
2. Click the **Clone** button. Cloning a blend creates a new blend with the exact same properties as the original.

6.2.4 User Property Tab

Refer to [Section 3.7 - User Property](#) in the [UniSim Design Simulation Basis Guide](#) for additional information.

Figure 6.4



A user property is any property that can be defined and calculated on the basis of composition. Examples for oils include RON and Sulfur content.

During the characterization process, all hypocomponents are assigned an appropriate property value. The value of the property for any flowsheet stream is then calculated. This enables user properties to be used as Column specifications.

Adding a User Property

1. Click the **Add** button. The User Property view appears.
2. Click the **Data** tab.
3. Use the drop-down list in the **Mixing Basis** field to select the basis for mixing.
4. Use the drop-down list in the **Mixing Rule** field to specify the mixing rule to use with your user property.
5. Specify values for the mixing rule parameters F1 and F2. By default these parameters have a value of 1.00.
6. Select a unit from the **Unit Type** drop-down list for the user property value. If the unit type is Temperature, the internal unit used is degrees Kelvin. (This is important when calculating the mixed value for a stream.)

7. Specify a lower and upper limit for your user property in the **Lower Limit Value** and **Upper Limit Value** cells.

The choice of Mixing Basis applies only to the basis that is used for calculating the property in a stream. You supply the property curve information on the same basis as the Boiling Point Curve for your assay.

The values you provide for the light end components are used when calculating the property value for each hypocomponent (removing that portion of the property curve attributable to the light ends components).

This is not the property curve information. These values determine how the user property is calculated in all flowsheet streams. When the value of a user property is requested for a stream, the composition in the specified basis is used and the property value is calculated using your mixing rule and parameters.

Editing a User Property

1. From the list of available user properties, click the user property you want to edit.
2. Click the **View** button. The User Property view appears.
3. Click the **Data** tab and modify the data for the user property.

Deleting a User Property

1. From the list of available user properties, click the user property you want to delete.
2. Click the **Delete** button.

You will not be prompted to confirm the deletion of a user property, so ensure you have selected the correct one before deleting.

Cloning a User Property

1. From the list of available user properties, select the user property you want to copy.
2. Click the **Clone** button. Cloning a user property creates a new user property with the exact same properties as the original.

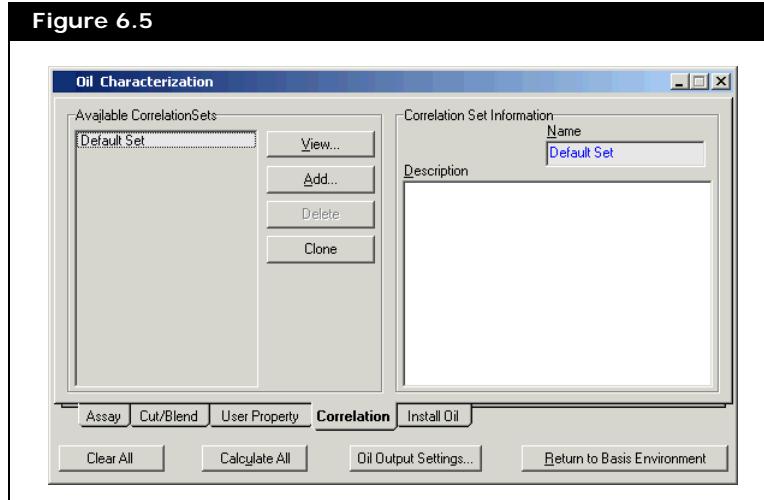
See [Section 3.8.1 - Correlation Tab](#) in the [UniSim Design Simulation Basis Guide](#) for more information.

6.2.5 Correlation Tab

This tab allows you to manage the correlations in a case. Use this tab to

add new correlation sets and modify existing sets.

Figure 6.5



You cannot change the correlations or ranges for the Default correlation set. If you want to specify different correlations or temperature ranges, create a new correlation set.

Adding a Correlation Set

1. Click the **Add** button. The Correlation Set view appears.
2. Click the **Options** tab and use the drop-down list in the MW column to specify the correlation you want to use.
3. Repeat step 2 for each of the columns in the table: SG, Tc, Pc, Acc. Factor and Ideal H.
Property correlations can be changed for the entire range or they can be made valid for only certain boiling point ranges.
4. To divide correlations over several boiling ranges, click the **Add New Range** button. The Add Temperature Range view appears.
5. In the **New Temp** field, enter the temperature at which you want to split the range.
6. Click the **Split Range** button and specify correlations in these two ranges. You can add as many temperature splits as required.
7. In the Assay and Blend Association group, check the **New Assays/Blends** checkbox to add all new assays and blends that were created using this correlation set.
8. Select the **Available Assays** radio button.
9. In the Assay/Blend table, select the **Use this Set** checkbox to use all the assays in this correlation set.
10. Select the **Available Blends** radio button.

11. In the Assay/Blend table, select the **Use this Set** checkbox to use all the blends in this correlation set.

Changes to the Molecular Weight or Specific Gravity correlations are applied to the assay curve, while the critical temperature, critical pressure, acentric factor and heat capacity correlations are applied to the blend's hypocomponent properties.

Changes to the assay correlations have no effect when you supply a property curve (e.g., Molecular Weight). The changes only apply when properties are being estimated.

Removing a Split

1. Click the **Remove Range** button. The Remove view appears.
2. From the list of available splits, select the split you want to remove.
3. Click the **Merge Temp Range** button. When you merge a range, you delete the correlations for the range with a Low End Temperature that is equal to the range temperature you are merging.

Changes to the correlations for an input assay results in the recalculation of that assay followed by the recalculation of any blend that uses it. Existing oil is automatically recalculated/re-cut using the new correlations and components installed in the flowsheet.

Editing a Correlation Set

1. From the list of available correlation sets, click the correlation set you want to edit.
2. Click the **View** button. The Correlation Set view appears.
3. Click the **Options** tab and modify the data that makes up the correlation set.

Deleting a Correlation Set

1. From the list of available correlation sets, click the correlation set you want to delete.
2. Click the **Delete** button.

You will not be prompted to confirm the deletion of a correlation set, so ensure the correct correlation set is selected before deleting.

Cloning a Correlation Set

1. From the list of available correlation sets, select the correlation set you want to copy.
 2. Click the **Clone** button. Cloning a correlation set creates a new correlation set with the exact same properties as the original.

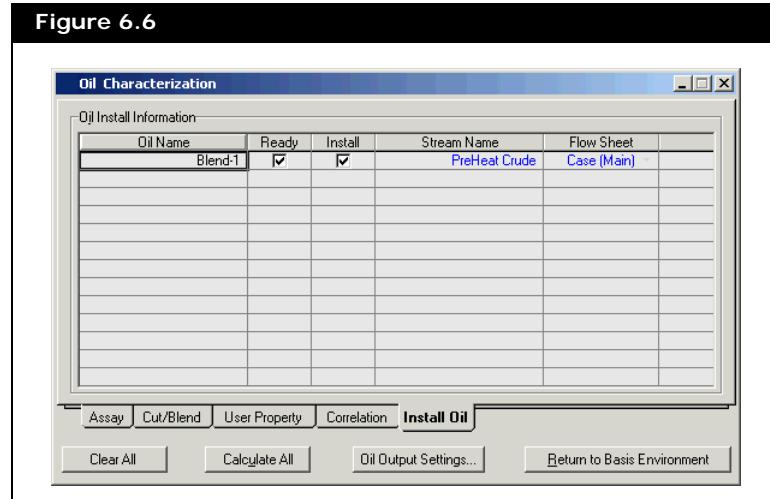
6.2.6 Install Oil Tab

See [Section 3.8.3 - Install Oil Tab](#) in the UniSim Design Simulation Basis Guide for more information.

Use this table to install an oil blend into a flowsheet.

1. Check the **Install** checkbox.
 2. In the Stream Name column, type a stream name for that blend. You can use an existing stream name or provide a new one. If you do not provide a name, the hypocomponents are not attached to the fluid package.
 3. Use the drop-down list in the Flow Sheet column to specify for which flowsheet you want to install the stream containing the hypocomponents.

Figure 6.6



7 Simulation Environment

7.1 Introduction	5
7.2 Main Properties	5
7.2.1 Status Message Tab.....	6
7.2.2 Calculation Levels Tab.....	6
7.2.3 Notes Tab	7
7.2.4 PV Balance Tab	7
7.2.5 Lock Tab	8
7.2.7 Converting a Case to a Template.....	9
7.3 UniSim Design XML.....	10
7.4 Optimizer.....	12
7.5 Event Scheduler.....	13
7.5.1 Adding a Schedule	14
7.5.2 Adding a Sequence.....	14
7.5.3 Adding an Event	15
7.5.4 Editing a Sequence.....	17
7.5.5 Editing an Event	18
7.5.6 Deleting a Schedule	19
7.5.7 Deleting a Sequence.....	19
7.5.8 Deleting an Event	20
7.5.9 Copying a Schedule.....	20
7.5.10 Copying a Sequence	20
7.5.11 Copying an Event	21
7.5.12 Importing a Schedule	21
7.5.13 Importing a Sequence.....	22
7.5.14 Exporting a Schedule	23
7.5.15 Exporting a Sequence	23
7.5.16 Sorting a Schedule	24
7.5.17 Sorting a Sequence	25
7.5.18 Sorting an Event	25
7.6 Integrator.....	26
7.6.1 Integration Time Group.....	27
7.6.2 Integration Step Size Group	28
7.7 Adjust-Recycle Manager	28
7.7.1 Simultaneous Adjust-Recycle Group.....	30

7.8 Initialize From	37
7.8.1 Terminology	38
7.8.2 Dynamic Usage Notes.....	39
7.8.3 Steady-State Usage Notes.....	49
7.8.4 Warning Messages	49
7.9 Dynamic/Steady State Modes.....	50
7.10 Solver Active/Holding.....	51
7.11 Integrator Active/Holding	51
7.12 Equation Summary	52
7.13 Enter Basis Environment.....	52
7.14 User Variables	52
7.14.1 Adding a User Variable	54
7.14.2 Editing a User Variable	54
7.14.3 Deleting a User Variable	54
7.15 Importing & Exporting User Variables	54
7.15.1 Importing User Variables.....	55
7.15.2 Exporting User Variables	55
7.16 Oil Output Settings	55
7.17 Object Navigator.....	56
7.17.1 Locating an Object	57
7.18 Simulation Navigator	58
7.18.1 Viewing an Object	59
7.19 Notes Manager.....	60
7.20 Optimization Objects	61
7.20.1 Adding an Optimization Object	62
7.20.2 Editing an Optimization Object	62
7.20.3 Deleting an Optimization Object	63
7.21 Reaction Package	63
7.22 Fluid Package/Dynamics Model	63
7.23 Workbook	64
7.23.1 Opening a Workbook.....	65
7.23.2 Installing Streams or Operations	67
7.23.3 Deleting Streams or Operations.....	67
7.23.4 Accessing Streams or Operations.....	68

7.23.5 Managing Workbook Tabs	71
7.23.6 Sorting Information	74
7.23.7 Exporting/Importing Workbook Tabs	76
7.24 PFD	77
7.24.1 Custom PFD Notebook	78
7.24.2 Locating Objects in PFD.....	79
7.24.3 Flowsheet Analysis Using the PFD	80
7.24.4 Access Column or Sub-Flowsheet PFDs.....	82
7.24.5 Opening Controller Face Plates	83
7.24.6 PFD Colour Schemes.....	84
7.24.7 Column Tray Section Display.....	87
7.24.8 PFD Tables	88
7.24.9 Multi-Pane PFDs	92
7.24.10 Exchanging XML Files	94
7.25 Column	94
7.26 Utilities	94
7.26.1 Adding Utilities.....	95
7.26.2 Viewing Utilities	97
7.26.3 Deleting Utilities.....	97
7.27 Simulation Balance Tool.....	98
7.27.1 Accessing the Simulation Balance Tool.....	98
7.27.2 Setup Tab.....	99
7.27.3 Summary Tab	100
7.27.4 Feeds/Products Tab	102
7.27.5 Transitions Tab	103
7.27.6 Adjust/Recycle Tab	105
7.27.7 Alerts Tab.....	106
7.27.8 Status Bar Monitoring	107

7.1 Introduction

Before entering the Simulation environment, you must have a fluid package with selected components in the component list and a property package.

If you do not have the above requirements, you cannot enter the Simulation environment.

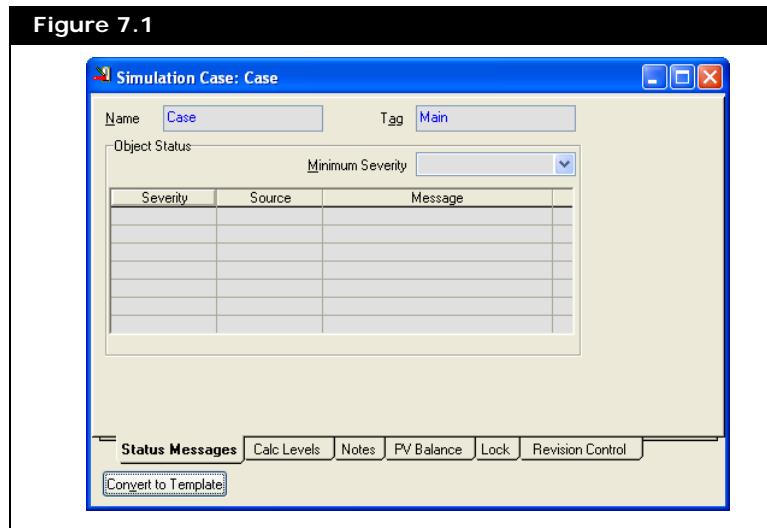
The Simulation environment contains the main flowsheet where you do the majority of your work (installing and defining streams, unit operations, columns and sub-flowsheets).

This flowsheet serves as the base level or “main” flowsheet for the whole simulation case. Any number of sub-flowsheets can be generated from the main flowsheet, but there is only one main flowsheet environment. Each individual sub-flowsheet that is installed has its own corresponding sub-flowsheet environment.

To enter the Simulation environment, click either the Enter Simulation Environment button or Return to Simulation Environment button in the Basis environment.

7.2 Main Properties

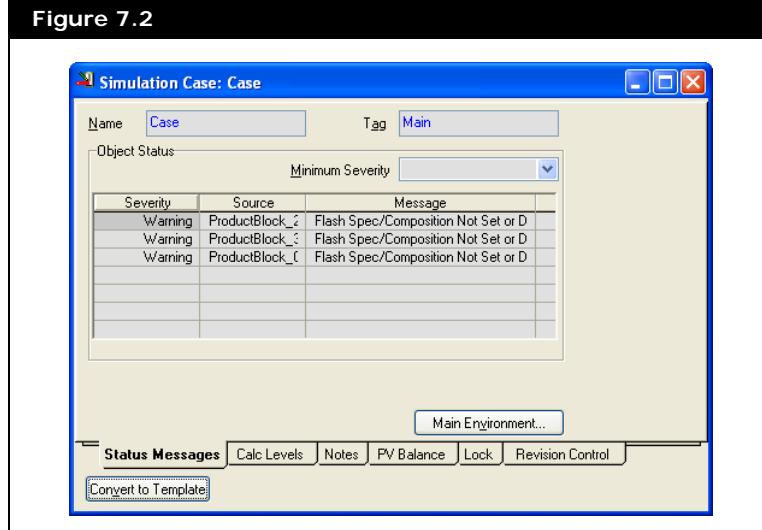
To access the Simulation Case view, select Main Properties from the Simulation menu, or use the **CTRL M** hot key combination. The Simulation Case view appears as shown below:



If Main Properties is selected in a sub-flowsheet environment, a Main Environment button is available for returning to the main environment.

7.2.1 Status Message Tab

The Status Message tab allows you to view any errors or warnings in your case, name the flowsheet, and provide a tag for it.



- To name the flowsheet, click the Name field and type a name for the flowsheet. The default name is **Case**.
- To add a tag to the flowsheet, click the Tag field and type a tag for your flowsheet. The default tag for the main flowsheet is **Main**.

The Object Status group displays the current status messages for all objects in the flowsheet according to the minimum severity. In the Minimum Severity drop-down list, click one of the following options:

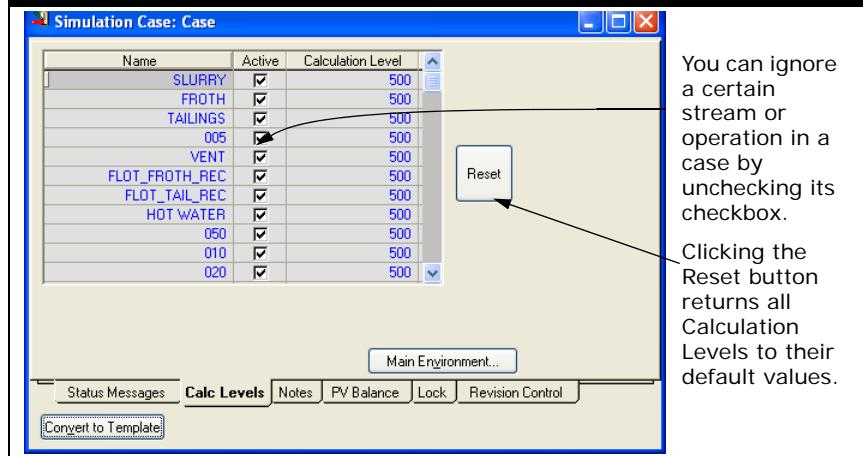
- **OK**. Sets the minimum severity at OK.
- **Optional Info**. Sets the minimum severity at Optional Info.
- **Warning**. Sets the minimum severity at Warning.
- **Required Info**. Sets the minimum severity at Required Info.
- ****Error****. Sets the minimum severity at **Error**.

7.2.2 Calculation Levels Tab

The Calc Levels tab controls the order in which streams, operations,

and flowsheets are calculated.

Figure 7.3



Use this procedure to change the calculation order/sequence of a separator in the PFD.

1. Open the UniSim Design case that contains the PFD you want to modify.
2. Enter the Simulation Environment.
3. From the **Simulation** menu, select the **Main Properties** command. The Simulation Case view will appear.
4. Click the **Calc Levels** tab in the Simulation Case view.
5. Locate the name of the separator in the table on the **Calc Levels** tab.
6. Enter a lower calculation level value in the appropriate cell under the **Calculation Level** column.

7.2.3 Notes Tab

To see all notes entered in the case, refer to [Section 7.19 - Notes Manager](#).

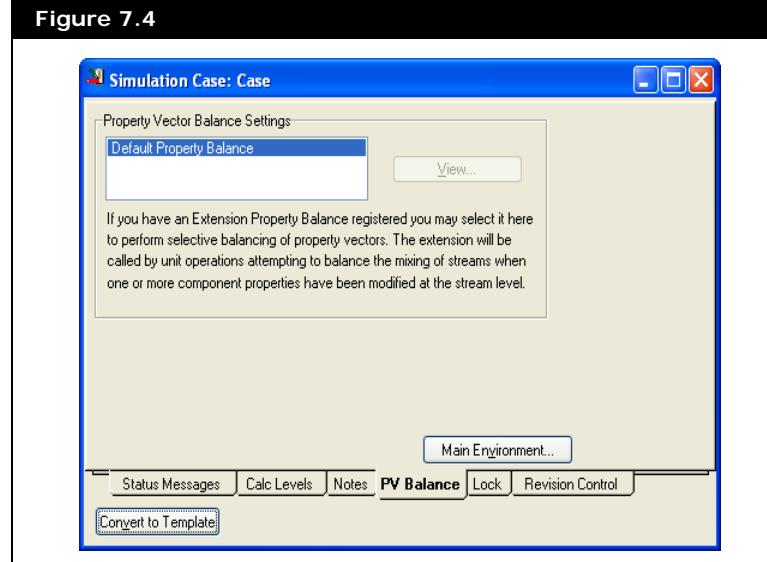
Use the Notes tab to add comments or descriptions about your simulation. For example, this is a good location for documenting changes made to the settings.

7.2.4 PV Balance Tab

The PV Balance tab enables you to modify the property vector balance

calculation.

Figure 7.4



There are two balance option available for you to select: Default Property Balance and Petroleum Assay Balance.

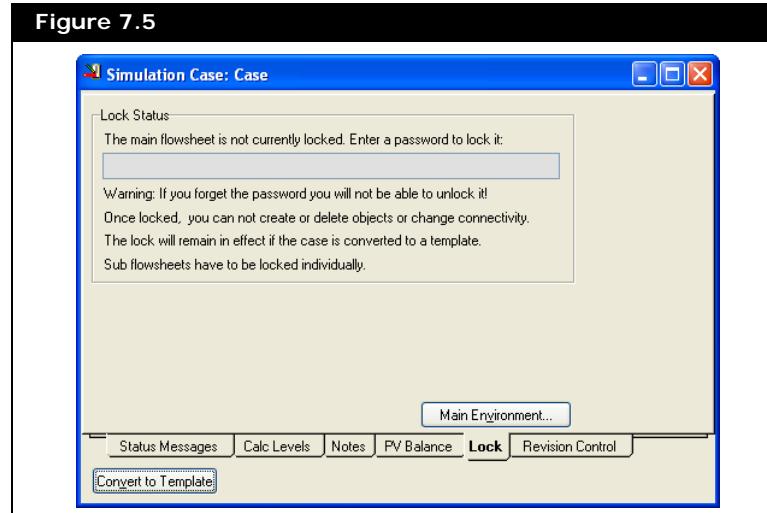
For the Petroleum Assay Balance, you can click the **View** button to modify the balance calculation method for the assay properties.

7.2.5 Lock Tab

Refer to [Section 11.15 - Case Security](#) for more information on locking flowsheet.

The Lock tab enables you to lock the **Main** flowsheet by entering a password in the field and press **ENTER**.

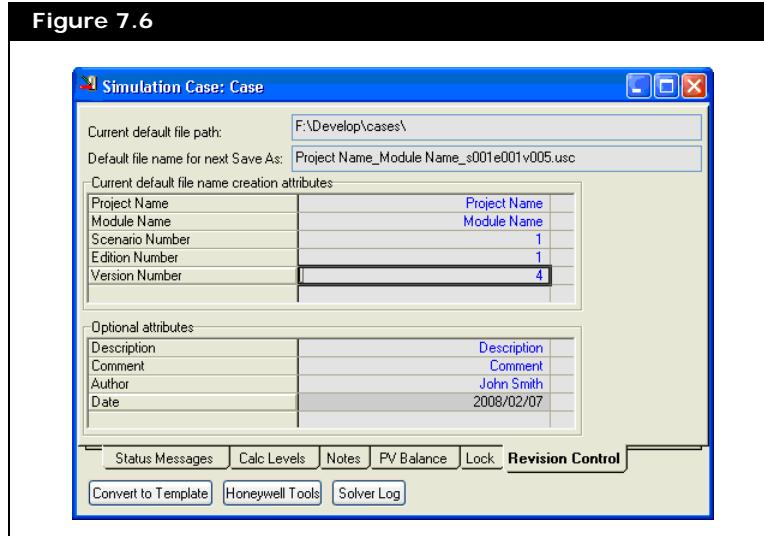
Figure 7.5



7.2.6 Revision Control Tab

The Revision Control Tab allows you to view or change the revision control information for the simulation case.

Figure 7.6



There are two groups in this view: **Current default file name creation attributes** and **Optional attributes**. The attributes in the former group are used to automatically create file names when the **Use revision control setting to create default file name for Save As** checkbox is checked in Revision Control page under Preferences/Files. The attributes in the latter group are used to enter some optional information for the case. The number of attributes, the name, data type of the attributes and the way to formulate the default file names can all be customized on the preferences page.

7.2.7 Converting a Case to a Template

Refer to [Section 3.5 - Templates](#) for additional information about templates and extra tabs.

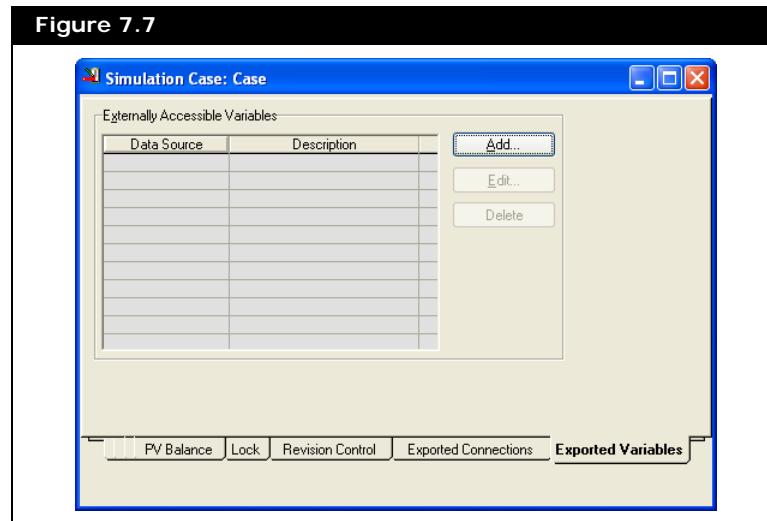
Click the Convert to Template button in the Simulation Case view. You are prompted to confirm your selection and save your case.

Converting your case to a template treats the flowsheet like a "black box" and allows you to install it as a sub-flowsheet operation in other cases.

After you convert the case to a template, the Exported Connections and Exported Variables tabs become available and the Convert to Template

button disappears from this view.

Figure 7.7

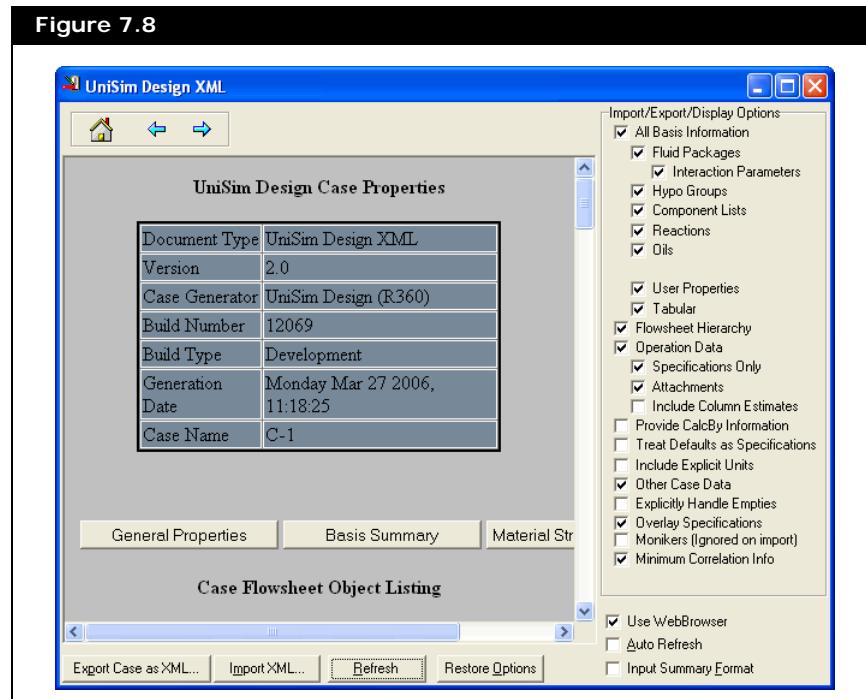


7.3 UniSim Design XML

UniSim Design now contains the ability to represent a simulation case in XML form. XML (Extended Markup Language) is used extensively in the computer software industry to facilitate data exchange between applications.

The UniSim Design XML view lists all the current specifications used to build the simulation case. To access the UniSim Design XML view:

1. From the **Simulation** menu, select **UniSim Design XML**.
2. The UniSim Design XML view appears.



Options	Description
	Allows you to return to the main Web page as shown in Figure 7.8 .
	Allows you to return to the previous page.
	Allows you to go to the next page.
Import/Export/Display Options	Use the check boxes to select the information you want to appear in the Datasheet.
Use WebBrowser	If you want to view the data in a web page, check the Use WebBrowser check box. If the check box is clear, XML code is displayed.
Auto Refresh	To automatically update the web page, check the Auto Refresh check box.

Options	Description
Input Summary Format	If you want to view all the user specifications in the UniSim Design v2.4 case, check the Input Summary Format check box. The functionality is similar to the Print-Specsheet-Flowsheet capability in UniSim Design 2.4.1, but reports more specifications to produce a more accurate representation of the case.
	Scroll up and down for a list of all flowsheet specification details. Click the buttons in the list to jump to the list displayed in groups.
Export Case as XML	Allows you to save the flowsheet specification details to an *.xml file.
Import XML	Allows you to open the flowsheet specification details from an *.xml file.
Refresh	Updates the list of current specifications in the flowsheet.
Restore	Restores the default import/export options.

There are a number of options associated with the UniSim Design XML output. These control the amount of information saved to the XML file.

Benefits of the UniSim Design XML tool:

- A structured output of simulation case data that can be queried using XML tools.
- An alternate form of case storage that allows the user to rebuild the case from the XML file.
- The ability to read partial information (additional pieces of equipment/streams or changed parameters for existing streams or operations) over top of an existing case.

Print the information in the Datasheet to an XML file:

1. Click the **Export Case as XML** button. The Save File view appears.
2. Specify a name and location for the Datasheet file, then click **Save**.

Import a Datasheet from an XML file:

1. Click the **Import XML** button. The Open File view appears.
2. Browse to the location of the required Datasheet file (*.xml).
3. Select the file you want to import, then click **Open**.

Refer to [Chapter 13 - Optimizer](#) in the [UniSim Design Operations Guide](#) for more information about the Optimizer.

7.4 Optimizer

The Optimizer performs steady state optimization by finding values of process variables that minimize or maximize a user-defined objective function. It has its own spreadsheet with attached variables that define

the objective function and mathematical expression relating to the variables.

In addition, you must set upper and lower bounds and constraints that the process variables must satisfy. The optimization must begin in a feasible region (i.e., all constraints must be satisfied at the initial starting conditions).

7.5 Event Scheduler

Refer to [Chapter 2 - Dynamic Tools](#) in the **UniSim Design Dynamic Modeling Guide** for additional information.

You can access the Event Scheduler using the **CTRL E** hot key combination.

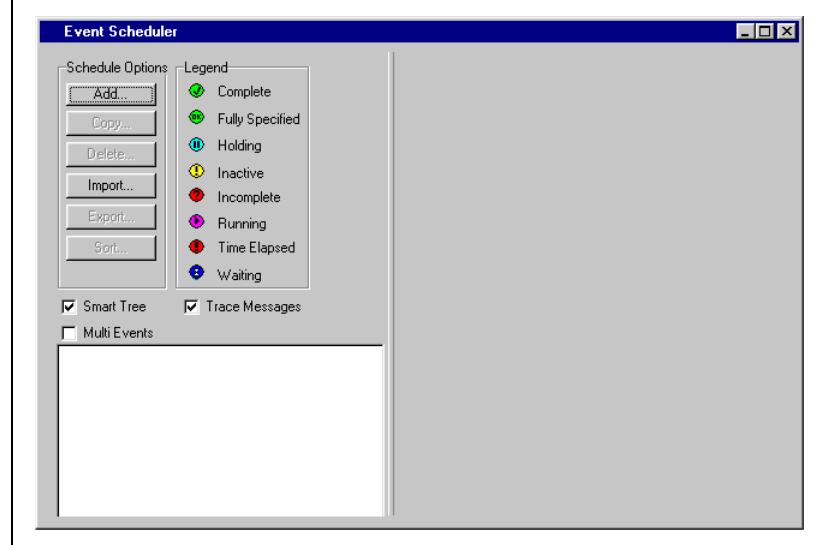
Use the Event Scheduler to perform tasks at specified times during the dynamic simulation of a process. Task times can be predetermined, or they can depend on the simulation.

For example, you can set a task to begin 20 minutes into the simulation or after a reboiler product stream temperature stabilizes.

To access the Event Scheduler view:

1. From the **Simulation** menu, select **Event Scheduler**.
2. The Event Scheduler view appears.

Figure 7.9



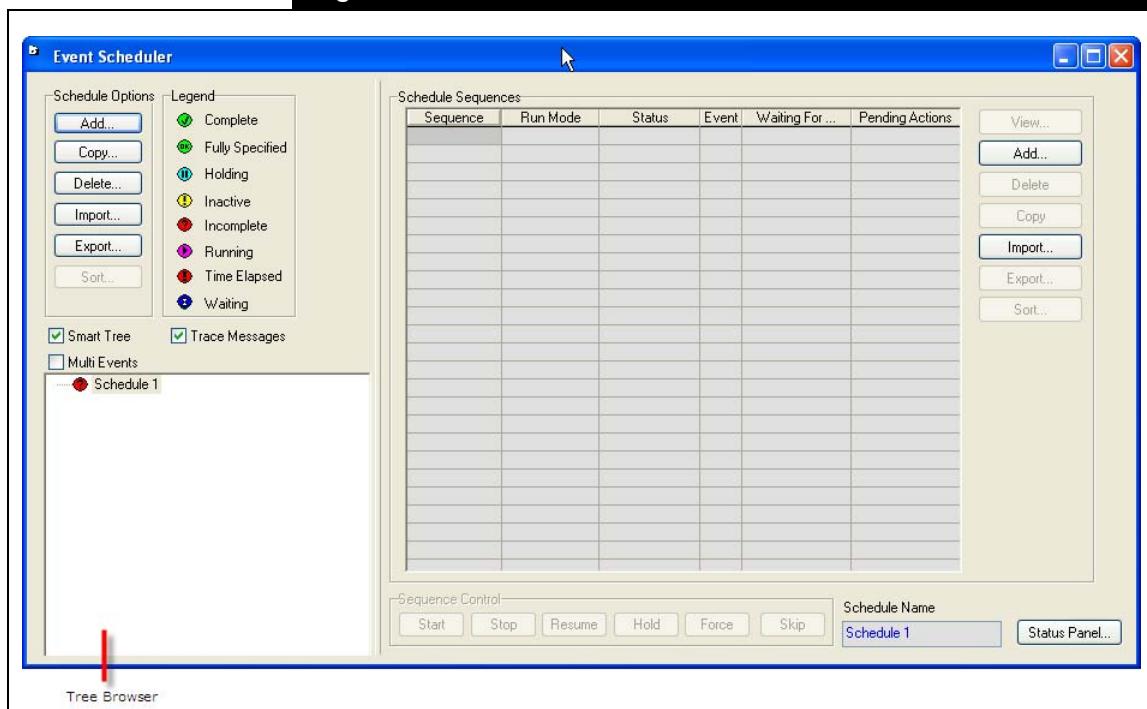
The Event Scheduler can be used only in Dynamics mode.

7.5.1 Adding a Schedule

To add a new schedule:

1. From the Event Scheduler view, click the **Add** button in the Schedule Options group.
2. The Schedule Sequences group appears.

Figure 7.10



7.5.2 Adding a Sequence

You can add as many sequences as needed in a schedule.

To add a new sequence:

1. From the Event Scheduler view, click the **Add** button in the

Schedule Options group. The Schedule Sequences group appears.

2. From the Schedule Sequences group, click the **Add** button. The sequence appears in the Schedule Sequences table.

Figure 7.11

Sequence	Run Mode	Status	Event	Waiting For...	Pending Actions
Sequence A	OneShot	Incomplete			

Sequence Control: Start, Stop, Resume, Hold, Force, Skip

Schedule Name: Schedule 1

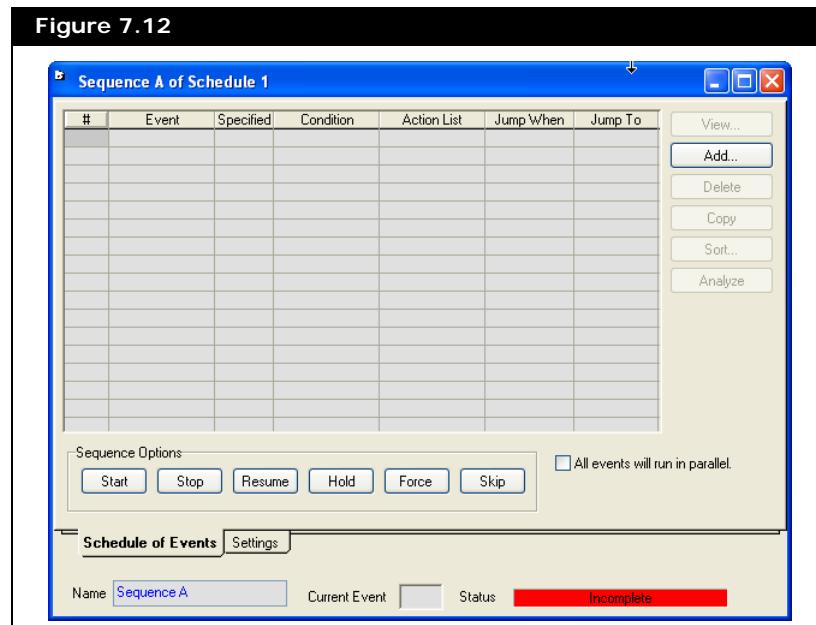
7.5.3 Adding an Event

You can add multiple events to a sequence.

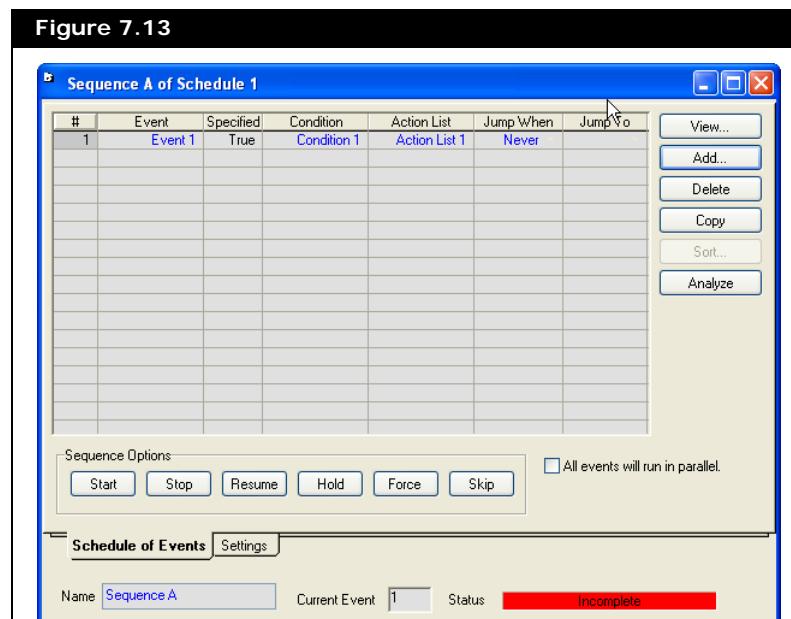
To add a new event to the sequence:

1. From the Event Scheduler view, click the **Add** button in the Schedule Options group. The Schedule Sequences group appears.
2. From the Schedule Sequences group, click the **Add** button. The sequence appears in the Schedule Sequences table.

- Click the **View** button in the Schedule Sequences group. The Sequence view appears.



- Click the **Add** button. The event appears in the table.

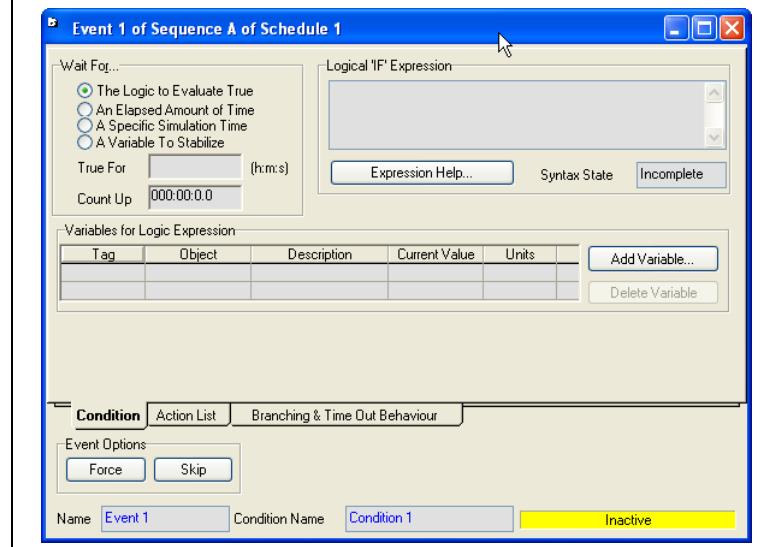


Defining an Event

- From the Sequence view, click the **View** button. The Event view

appears, which allows you to define events.

Figure 7.14



The parameters on the view change depending on the radio button you select. Refer to [Section 2.5.4 - Event View](#) in the **UniSim Design Dynamic Modeling Guide** for more information.

Refer to [Action List Tab](#) in [Section 2.5.4 - Event View](#) from the **UniSim Design Dynamic Modeling Guide** for more information.

2. Click on the **Condition** tab. In the Wait For group, select the condition you are waiting for before running the event. Select from the following options:
 - The Logic to Evaluate True
 - An Elapsed Amount of Time
 - A Specific Simulation Time
 - A Variable to Stabilize
3. Click the **Action List** tab.
4. Close the Event and Sequence views to return to the Event Scheduler view.
5. Click the **Start** button to start the integrator.

7.5.4 Editing a Sequence

1. From the Schedule Sequences group of the Event Scheduler view,

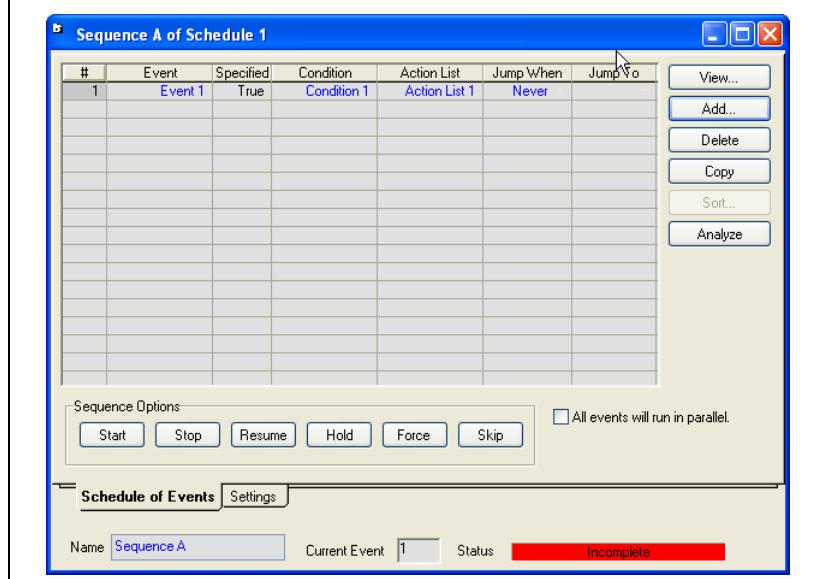
select the sequence you want to edit.

Figure 7.15

Schedule Sequences					
Sequence	Run Mode	Status	Event	Waiting For ...	Pending Actions
Sequence A	OneShot	Incomplete			
Sequence B	OneShot	Incomplete			

2. You can modify the sequence name and run mode. If you want to modify the information for the defined event and the sequence universal settings, click the **View** button. The Sequence view appears.

Figure 7.16



7.5.5 Editing an Event

1. From the Schedule Sequences group of the Event Scheduler view, select the sequence you want to edit the defined events.

Figure 7.17

Schedule Sequences					
Sequence	Run Mode	Status	Event	Waiting For ...	Pending Actions
Sequence A	OneShot	Incomplete			
Sequence B	OneShot	Incomplete			

2. Click the **View** button in the Schedule Sequences group. The Sequence view appears.

- From the Sequence view, select the event you want to edit.

Figure 7.18

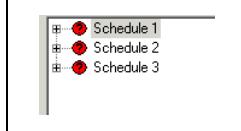
#	Event	Specified	Condition	Action List	Jump When	Jump To
1	Event 1	True	Condition 1	Action List 1	Never	
2	Event 2	True	Condition 2	Action List 2	Never	

- Click the **View** button. The Event view appears.
- Modify the event as required.

7.5.6 Deleting a Schedule

- From the Event Scheduler view, select the schedule you want to delete from the tree browser.

Figure 7.19



- Click the **Delete** button in the Schedule Options group.

You will not be prompted to confirm the deletion of the schedule, even if you have sequences and events defined. Ensure the correct schedule is selected before deleting.

7.5.7 Deleting a Sequence

- From the Schedule Sequences group of the Event Scheduler view, select the sequence you want to delete.

Figure 7.20

Schedule Sequences					
Sequence	Run Mode	Status	Event	Waiting For ...	Pending Actions
Sequence A	OneShot	Incomplete			
Sequence B	OneShot	Incomplete			

The Delete button is only active when a sequence exists in the case.

- Click the **Delete** button in the Schedule Sequences group.

You will not be prompted to confirm the deletion of the sequence, even if you have events defined. Ensure you selected the correct sequence before deleting.

The Delete button is only active when a sequence exists in the case.

7.5.8 Deleting an Event

- From the Schedule Sequences group of the Event Scheduler view, select the sequence you want to delete the defined events.

Figure 7.21

Sequence	Run Mode	Status	Event	Waiting For...	Pending Actions
Sequence A	OneShot	Incomplete			
Sequence B	OneShot	Incomplete			

- Click the **View** button. The Sequence view appears.

Figure 7.22

#	Event	Specified	Condition	Action List	Jump When	Jump To
1	Event 1	True	Condition 1	Action List 1	Never	
2	Event 2	True	Condition 2	Action List 2	Never	

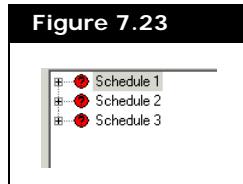
The Delete button is only active when an event exists in the sequence.

- From the Sequence view, select the event you want to delete, and click the **Delete** button.

You will not be prompted to confirm the deletion of the event, so ensure you selected the correct event before deleting.

7.5.9 Copying a Schedule

- From the Event Scheduler view, select the schedule you want to copy from the tree browser.



The Copy button is only active when a schedule exists in the case.

- Click the **Copy** button in the Schedule Options group to make a copy of the selected schedule. The copied schedule is added to the tree browser.

7.5.10 Copying a Sequence

The Copy button is active when a sequence exists in the schedule.

- From the Schedule Sequences group of the Event Scheduler view,

select the sequence you want to copy.

Figure 7.24

Schedule Sequences					
Sequence	Run Mode	Status	Event	Waiting For ...	Pending Actions
Sequence A	OneShot	Incomplete			
Sequence B	OneShot	Incomplete			

2. Click the **Copy** button in the Schedule Sequences group to make a copy of the selected sequence.

7.5.11 Copying an Event

1. From the Schedule Sequences group of the Event Scheduler view, select the sequence you want to copy the defined events.

Figure 7.25

Schedule Sequences					
Sequence	Run Mode	Status	Event	Waiting For ...	Pending Actions
Sequence A	OneShot	Incomplete			
Sequence B	OneShot	Incomplete			

2. Click the **View** button. The Sequence view appears.

Figure 7.26

#	Event	Specified	Condition	Action List	Jump When	Jump To
1	Event 1	True	Condition 1	Action List 1	Never	
2	Event 2	True	Condition 2	Action List 2	Never	

The Copy button is only active when an event exists in the sequence.

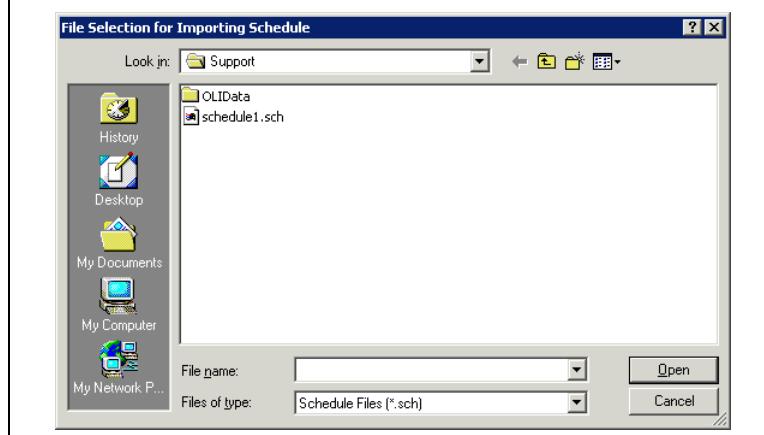
3. From the Sequence view, select the event you want to copy, and click the **Copy** button.

7.5.12 Importing a Schedule

1. From the Schedule Options group of the Event Scheduler view, click the **Import** button. The File Selection for Importing Schedule view

appears.

Figure 7.27

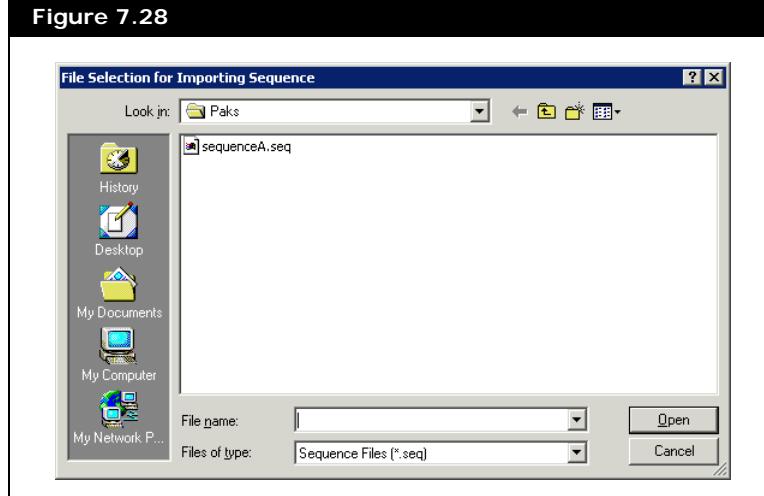


2. Browse to the location of the saved schedule file (*.sch).
3. Select the file you want to import, then click the **Open** button.

7.5.13 Importing a Sequence

1. From the Schedule Sequences group of the Event Scheduler view, click the **Import** button. The File Selection for Importing Sequence view appears.

Figure 7.28



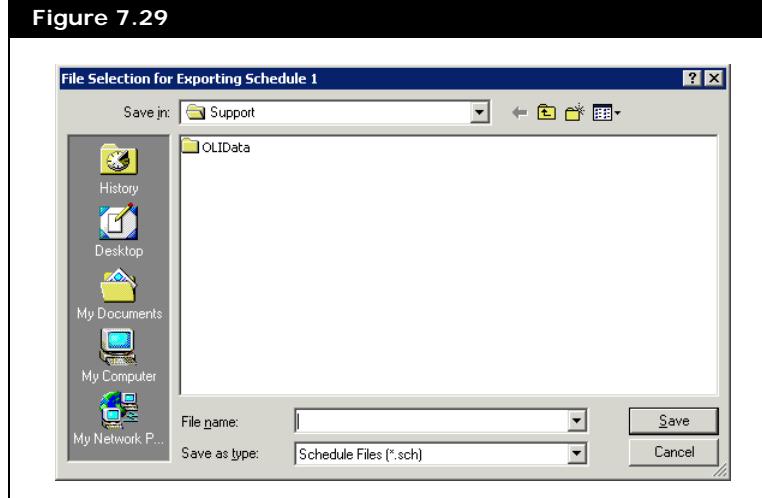
2. Browse to the location of the saved sequence file (*.seq).
3. Select the file you want to import and click the **Open** button.

7.5.14 Exporting a Schedule

1. From the Event Scheduler view, select the schedule you want to export from the tree browser.
2. Click the **Export** button. The File Selection for Exporting Schedule view appears.

The Export button is only active when a schedule exists in the case.

Figure 7.29



3. In the **File name** field, specify the name for the schedule file
4. From the **Save in** drop-down list, select the location to save the schedule file and then click the **Save** button.

7.5.15 Exporting a Sequence

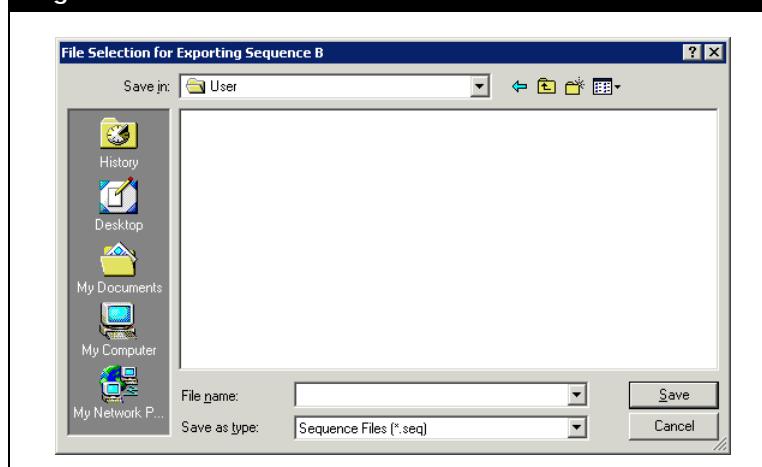
1. From the Schedule Sequences group of the Event Scheduler view,

The Export button is only active when a sequence exists in the schedule.

select the sequence you want to export.

2. Click the **Export** button. The File Selection for Exporting Sequence view appears.

Figure 7.30



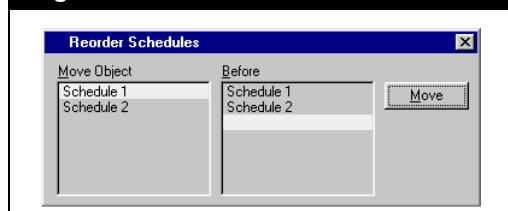
3. In the **File name** field, specify the name for the schedule file
4. From the **Save in** drop-down list, select the location to save the sequence file and then click the **Save** button.

7.5.16 Sorting a Schedule

The Sort button is only active when at least two schedules exist in the case.

1. From the Schedule Options group of the Event Scheduler view, click the **Sort** button. The Reorder Schedules view appears.

Figure 7.31

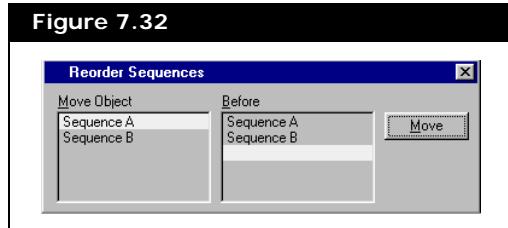


2. From the Move Object list, select the schedule you want to move.
3. From the Before list, select the schedule you want to insert the schedule you are moving before.
4. Click the **Move** button to complete the move.
5. Close the Reorder Schedules view when you have completed sorting the schedules.

To move a schedule to the end of the list, select the blank space under the last schedule in the Before list.

7.5.17 Sorting a Sequence

- From the Schedule Sequences group of the Event Scheduler view, click the **Sort** button. The Reorder Sequences view appears.



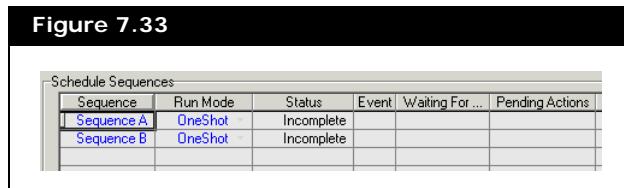
The Sort button is only active when at least two sequences exist in the schedule.

To move a sequence to the end of the list, select the blank space under the last sequence in the Before list.

- From the Move Object list, select the sequence you want to move.
- From the Before list, select the sequence you want to insert the sequence you are moving before.
- Click the **Move** button to complete the move.
- Close the Reorder Sequences view when you have completed sorting the sequences.

7.5.18 Sorting an Event

- From the Schedule Sequences group of the Event Scheduler view, select the sequence you want to sort the defined events.

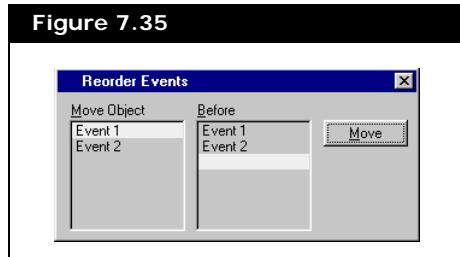


- Click the **View** button. The Sequence view appears.

Figure 7.34

#	Event	Specified	Condition	Action List	Jump When	Jump To
1	Event 1	True	Condition 1	Action List 1	Never	
2	Event 2	True	Condition 2	Action List 2	Never	

3. Click the **Sort** button. The Reorder Events view appears.



The Sort button is only active when at least two events exist in the sequence.

4. From the Move Object list, select the event you want to move.
5. From the Before list, select the event you want to insert the event you are moving before.
6. Click the **Move** button to complete the move.
7. Close the Reorder Events view when you have completed sorting the events.

7.6 Integrator

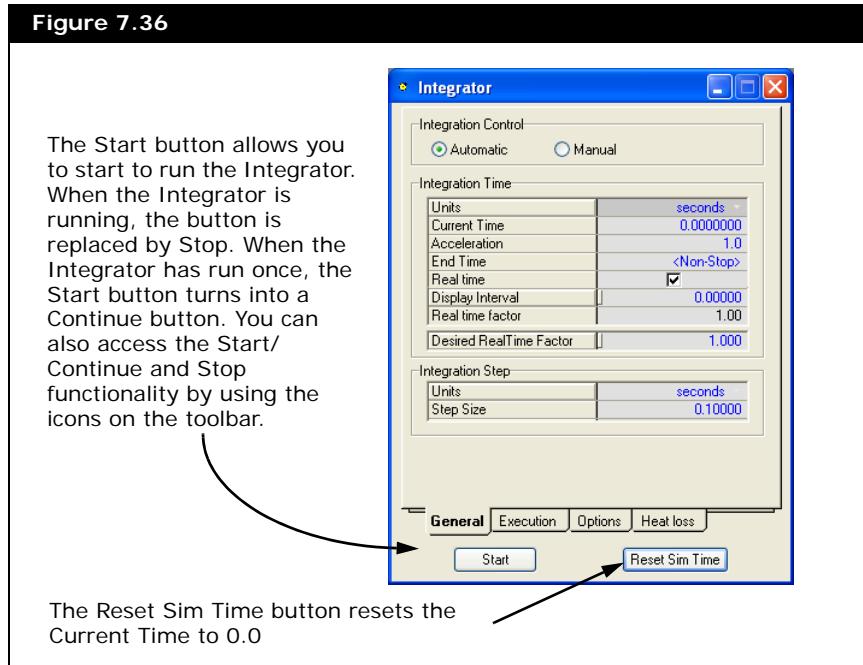
Refer to [Section 2.4 - Integrator](#) in the **UniSim Design Dynamic Modeling Guide** for additional information.

You can use the Integrator when running a case in Dynamic mode. There are two ways you can access the Integrator:

- From the **Simulation** menu, select **Integrator**.

- Press **CTRL I**.

Figure 7.36



7.6.1 Integration Time Group

The Integration Time group contains the following parameters:

Parameter	Description
Units	Time units for the Current Time, End Time, and Display Interval fields.
Current Time	Displays the time that the Integrator is running. When the Integrator is Reset, this value returns to zero. When the Integrator is not running, you can specify the value for the current time.
Acceleration	Actual Step Size = Acceleration * Specified Step Size. This always applies.
End Time	Allows you to specify the time at which the Integrator stops.
Real Time	Activates the Desired Real Time Factor field.
Display Interval	Visible only in Automatic Integration Control, this field contains the time interval at which UniSim Design updates the views. The frequency of updating has a significant impact on the speed at which your simulation runs. The Display Interval has no effect on the calculation frequency.
Real time factor	Visible only in Automatic Integration Control, this field is calculated by dividing a time interval for a case by the actual time required by UniSim Design to simulate that time interval. The Real time factor depends on the computer's processing speed and the complexity of the simulation case.

Parameter	Description
Desired Real Time Factor	Appears only when you check the Real time checkbox. Allows you to set the speed at which the integrator operates. The default setting of 1 indicates that the integrator is running at actual time, which appears on the status bar of the UniSim Design Desktop. You have the option to increase (>1) or decrease (<1) the speed of the integration. The Desired Real Time Factor has no effect on the calculation and results.
Number of time steps to execute:	Number of time steps at which UniSim Design executes. This field appears only when you click the Manual radio button in the Integration Control group.

7.6.2 Integration Step Size Group

The Integration Step Size group contains the following parameters:

Parameter	Description
Units	Allows you to select the units for the integration step size.
Step Size	Allows you to specify the integration step size, which by default is 0.5 seconds. While the integrator is running, this value cannot be changed.

7.7 Adjust-Recycle Manager

The Adjust-Recycle Manager (ARM) allows you to monitor and control all simultaneous groups in the case.

Access the Adjust-Recycle Manager by selecting the Adjust-Recycle Manager command from the Simulation menu.

The ARM contains the following tabs:

- Configuration
- Parameters

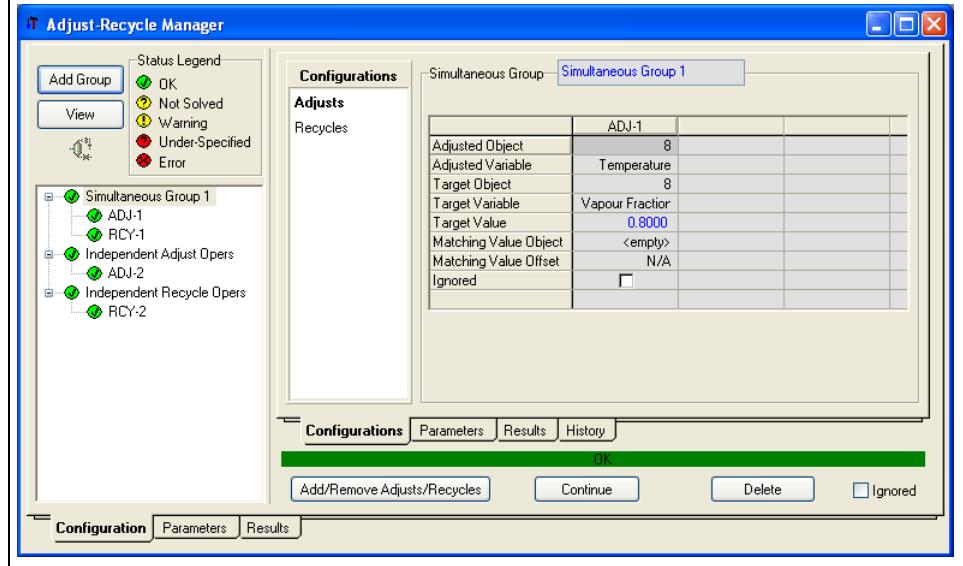
Configuration Tab

The Configuration tab allows the user to:

- add new simultaneous groups
- view selected simultaneous group
- display information regarding simultaneous groups,
- displays independent adjusts and independent recycles within the case, and

- locate selected adjusts and recycles on the PFD.

Figure 7.37



To add new simultaneous groups, just click button **Add Group**, a new simultaneous group will be created and be added to the list of groups available in the case.

To view an object, just select the object (e.g., simultaneous group, adjust or recycle), and click button **View**, then the view of the selected object will pop up.

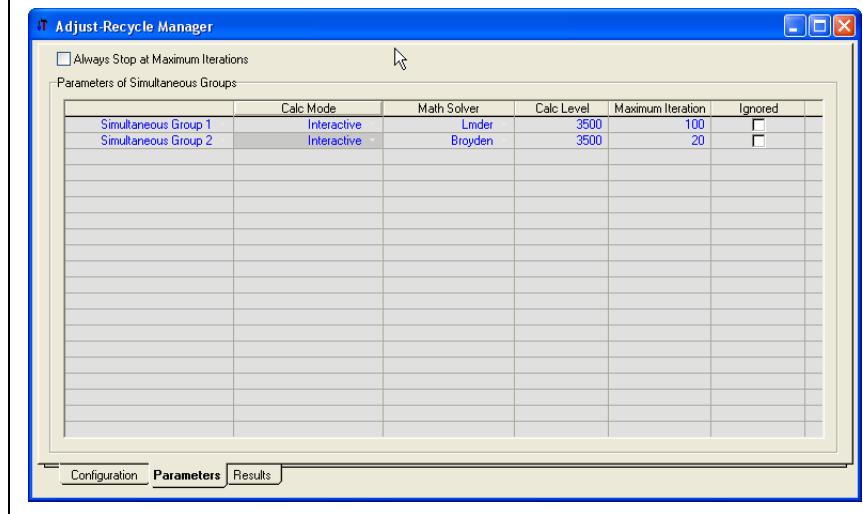
To locate selected adjust or recycle on PFD, select adjust or recycle in the list and click button **Locate Object in PFD**. The location of the selected object will pop up.

Parameters Tab

The Parameters tab allows the user to modify calculation mode, math solver, calculation level and maximum number of iterations of each simultaneous group. This tab also allows the user to ignore individual

simultaneous group.

Figure 7.38



Always Stop at Maximum Iterations If checked on, any Adjust operation in the case will stop as soon as the number of iterations exceeds the user specified **Maximum Iterations**. Otherwise, it will behave according to the specifications of **Always Stop at Max Iters** for the specific adjust.

7.7.1 Simultaneous Adjust-Recycle Group

The Simultaneous Adjust-Recycle Group (SARG) allows you to monitor and modify adjusts and recycles in the group. This gives you access to a more efficient calculation method with more control over the calculations.

The SARG requires at least two active (in other words, not ignored) adjusts or at least one active recycles to solve. If you are using only one adjust, you cannot use the SARG.

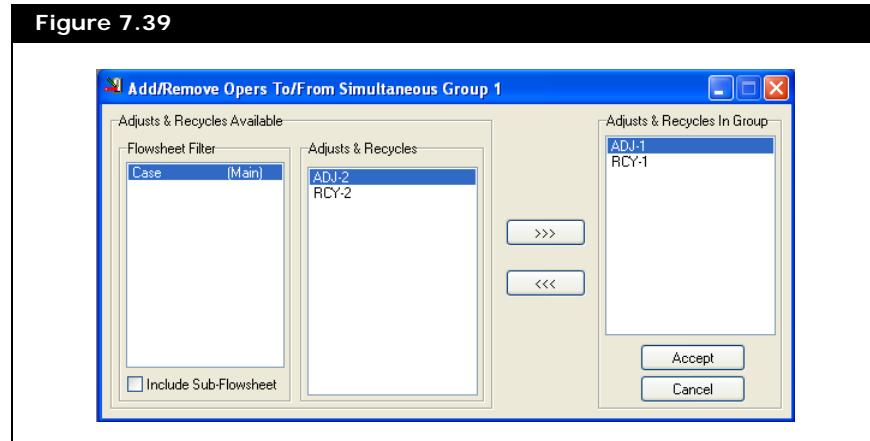
The ARG contains the following tabs:

- Configurations
- Parameters
- Results
- History

The SARG also consists of the Add/Remove Adjusts/Recycles, Continue

and Delete buttons. The Ignored checkbox when activated switches the SARG off, as well as all of adjusts and recycles in the group.

To add or remove adjusts and recycles from SARG, click button **Add/Remove Adjusts/Recycles**, and the following view will pop up.

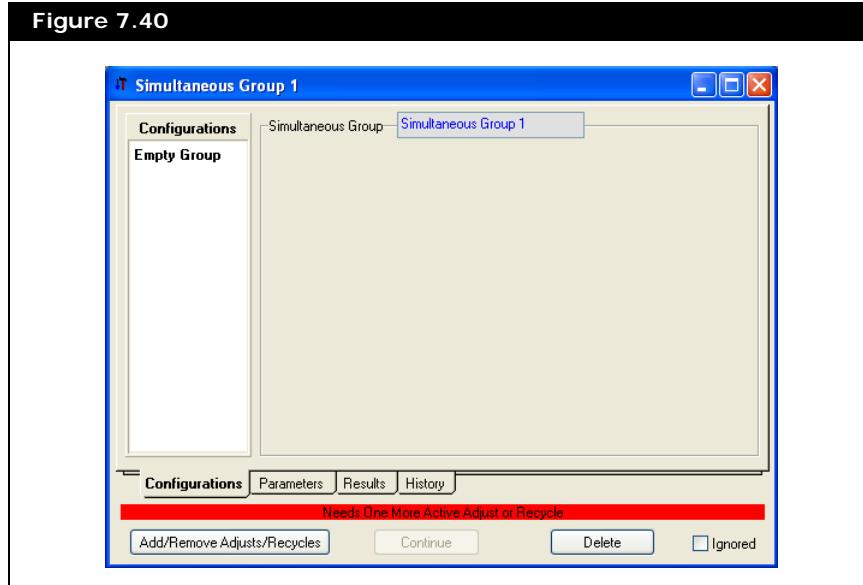


Procedures for Adding or Removing Adjusts and Recycles

1. Select the flowsheet containing adjusts and recycles you want from the FlowSheets Filter. If you want to include adjusts and recycles located in the sub-flowsheets of the selected flowsheet, active checkbox "Include Sub-Flowsheet".
2. Select the adjusts and recycles you want to add from the second column. Click the button **>>>** to add the selected objects to the column Adjusts and Recycles in Group.
You can remove objects from the column Adjusts & Recycles in Group by selecting the objects you do not want from the list, and clicking the button **<<<**.
3. Click button **Accept** to accept or click button **Cancel** to cancel the changes made in the above step.

When a new simultaneous group is created, before any adjust or recycle is added to the group, page Empty Group will be displayed on

each tab of the group.



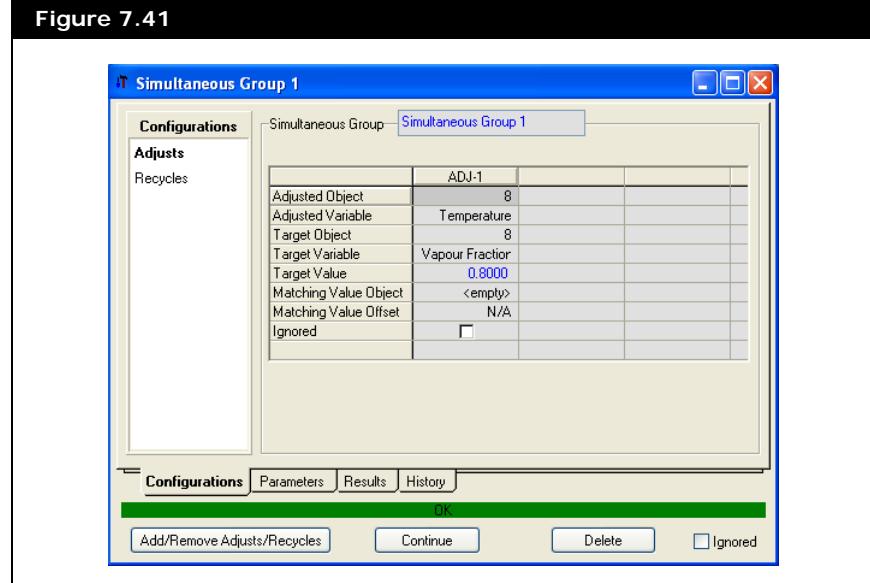
After adjusts and/or recycles are added to the group, page Empty Group will be removed. New page Adjusts if there is at least one adjust in the group, and new page Recycles if there is at least one recycle in the group, will be added.

Configuration Tab

The Configuration tab displays information regarding adjusts and recycles in the group on pages Adjusts and Recycles. You can view the individual adjusts or recycles by double-clicking on their names. You can modify the target value or matching value object, value, and offset of the adjusts, and specified conditions and compositions of inlet and outlet streams of the recycles. This tab also allows you to ignore

individual adjusts and recycles.

Figure 7.41



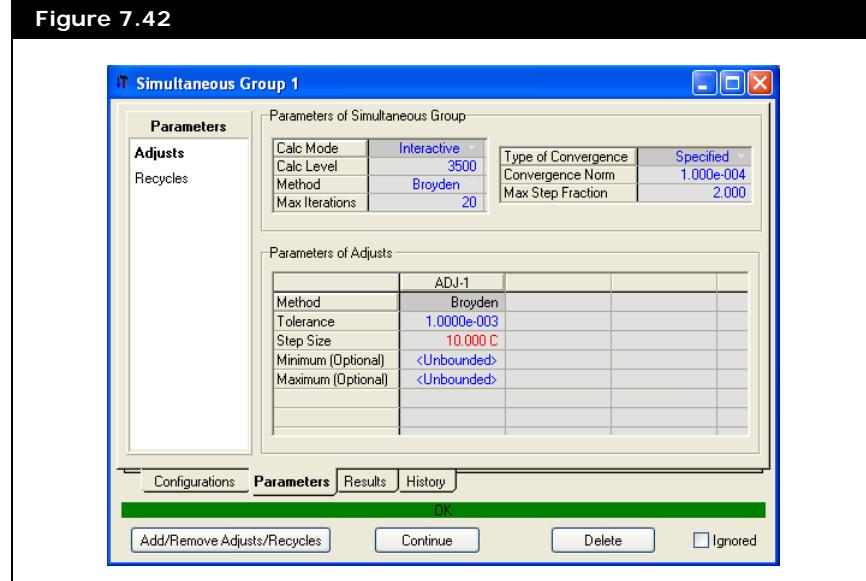
Parameters Tab

The Parameters tab allows you to modify the method, tolerance, step size, maximum iterations, max, and min values of each Adjust, to modify acceleration method, maximum iterations, flash type, acceleration frequency, Q maximum, Q minimum and acceleration delay of each recycle. This tab also allows you to specify some of the calculation parameters of the group as described in the table below.

Parameter	Description
Calc Mode	Allows you to select one of two simultaneous calculation modes: Interactive Adjusts/Recycles are solved simultaneously by solving a set of nonlinear equations using numerical methods such as Lmder and Broyden. Non-Interactive Adjusts/Recycles are solved at the same time using their own methods (e.g., Secant, Wegstein), and new calculated values are pushed back by the group to flowsheet simultaneously
Calc Level	Allows the user to control the order in which groups are calculated by flowsheet solvers within the case. The lower the calc level of a group, the earlier the group is solved. Adjusts and recycles in the simultaneous group will use Calc Level of the group. Calc Level of the adjusts and recycles themselves will not be applied.

Parameter	Description
Method	<p>Allows to select one of two math solvers when Interactive mode is applied:</p> <p>Broyden Use a unit matrix as initial Jacobian matrix, and apply Sherman-Morrison formula to update inverse of Jacobian matrix.</p> <p>Lmder It is a modification of the Levenberg-Marquardt algorithm, and needs derivatives calculation by perturbation at every iteration step. Lmder is not applied if the group includes recycles.</p>
Max # of Iterations	Maximum number of iterations for the group.
Type of Jacobian Calculations	<p>Allows you to select one of three Jacobian calculations:</p> <p>ResetJac Jacobian is fully calculated and values reset to initial values after each jacobian calculation step. Most time consuming but most accurate.</p> <p>Continuous Values are not recalculated between Jacobian calculation steps. Quickest, but allows for "drift" in the Jacobian therefore not as accurate.</p> <p>Hybrid Hybrid of the above two methods.</p>
Type of convergence	<p>Allows you to select one of three convergence types:</p> <p>Specified SARG is converged when all adjusts and recycles are within the specified tolerances.</p> <p>Norm SARG is converged when the norm of the residuals (sums of squares) is less than a user specified value.</p> <p>Either SARG is converged with whichever of the above types occurs first.</p>
Max Step Fraction	The number x step size is the maximum that the solver is allowed to move during a solve step.
Perturbation Factor	The number x range (Max - Min) or the number x 100 x step size (if no valid range). This is the maximum that the solver is allowed to move during a Jacobian step when Lmder is applied.

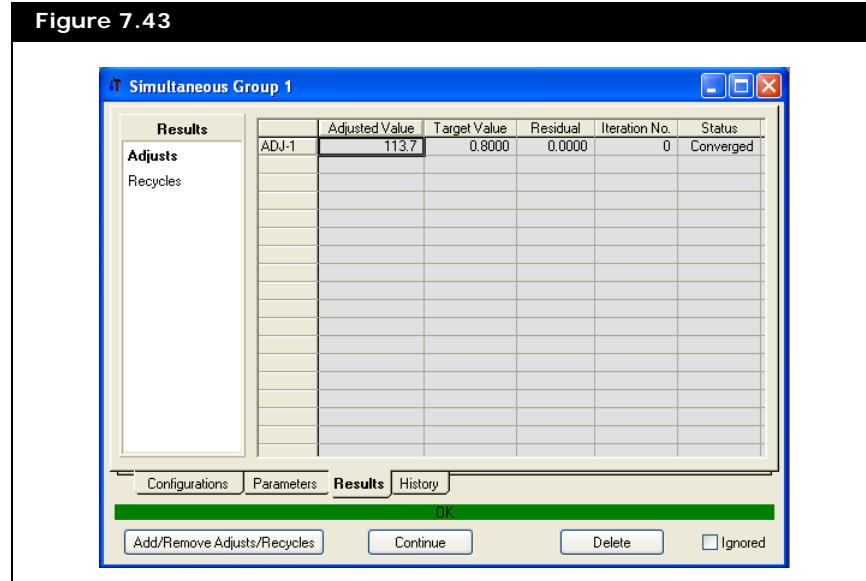
Figure 7.42



Results Tab

The Results tab displays the adjusted value, target value, residual value, iteration number and status of adjusts, as well as iteration number and status of recycles when solving of the group is finished.

Figure 7.43



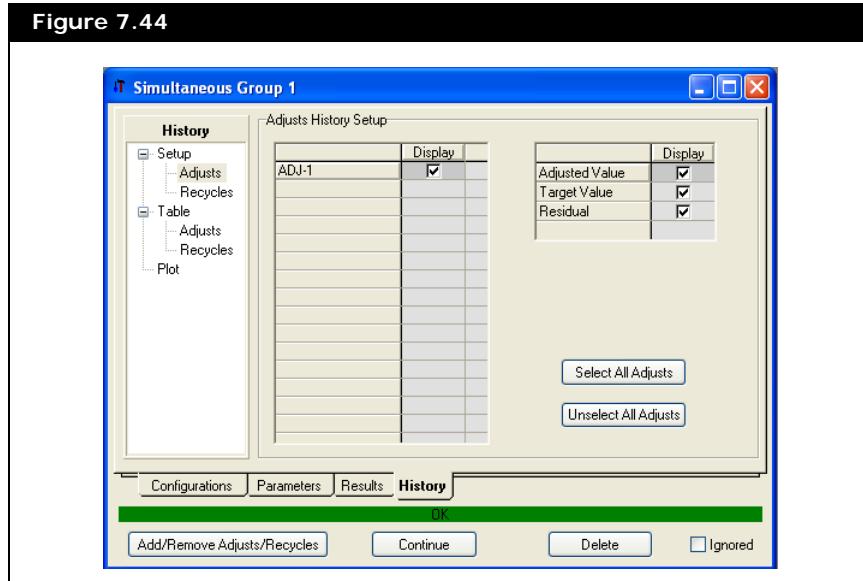
History Tab

The History tab contains the following three pages to monitor history of adjust and recycles for each iteration of the group.

- SetUp
 - Table
 - Plot

The SetUp page allows to specify which adjusts, recycles and variables you want to view or monitor. To view an adjust or a recycle, activate the Display checkbox beside the adjust or recycle name. Adjusts or recycles are always viewed in order from left to right across the page. For example, if you are viewing Adjust 2 and add Adjust 1 to the view, Adjust 1 becomes the first set of numbers, and Adjust 2 is shifted to the right. To view a variable, activate the Display checkbox corresponding to the variable of interest.

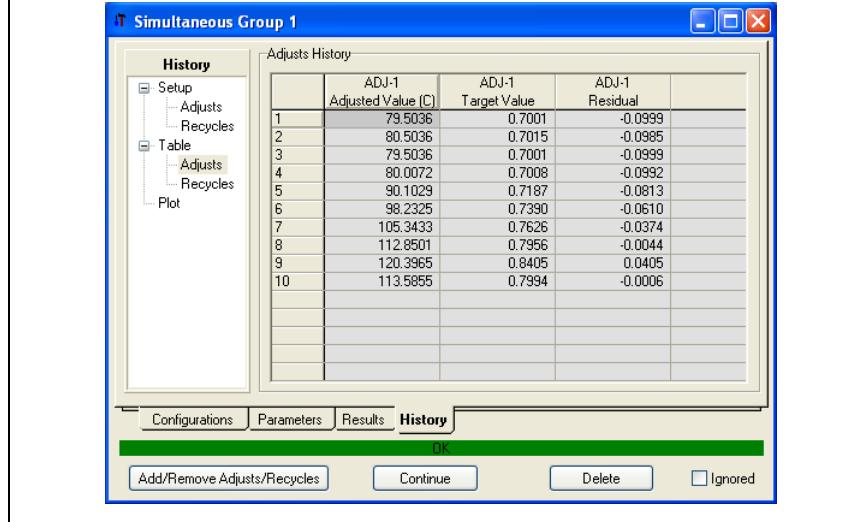
Figure 7.44



The Table page and Plot page display the convergence information of the selected variables corresponding to the selected adjusts and recycles as the calculations are performed in tabular and graphical format respectively.

The History tab only displays the values from a solve step. The values calculated during a Jacobian step can be seen on the Monitor tabs of adjusts and recycles for the individual results.

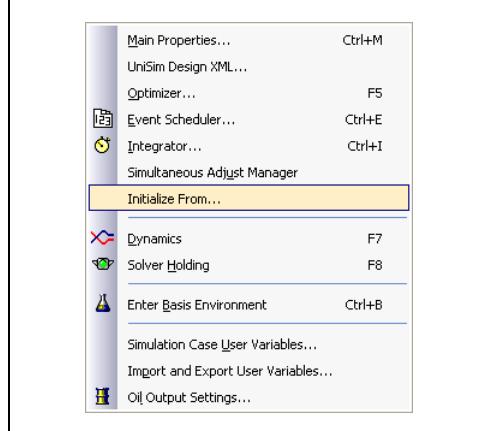
Figure 7.45



7.8 Initialize From

The Initialize From command is located in the Simulation menu and is used to initialize the dynamic data from another saved case into the currently opened case while leaving all of the static data unchanged.

Figure 7.46



Note that the Initialize From operation is used to initialize the entire flowsheet. If you want to only initialize a portion of the flowsheet, refer to [Section 2.7.2 - Initialize From](#) in the **Dynamic Modeling Reference Guide**.

DynamicInitializeFrom is available from the OLE interface.

7.8.1 Terminology

A typical project may have to maintain multiple cases, where each case represents different operating conditions. The underlying flow scheme and equipment details are the same for each case, but the operating conditions are different. For example, a project might need a case for: steady-state, shutdown, summer operation, maximum throughput, maximum diesel production, etc.

If the equipment details are changed (because of a plant debottlenecking project or update of design data from the subcontractor), then it will be necessary to change the equipment details for every maintained case. The manual method to maintain multiple cases would be to open each individual case and change the equipment sizes by hand. The Initialize From functionality allows you to make all of the equipment changes to one case, and then automatically update the other cases.

Some key terminology:

Case Management	A process undertaken by a simulation engineer to maintain a set of cases that model a plant at different operating conditions.
IC	Initial Condition. The starting operating conditions of a case. A case will sometimes be referred to as an IC.
IC Set	A set of initial conditions (cases) that model important operating regimes in the plant. Typically used as starting points for training exercises or to define design limitations for various pieces of equipment / unit operations.
Target Case	Master case (at any operating condition) that includes the superset of all unit operations and the most recent static initialization data
Source Case(s)	Secondary case(s) at different operating conditions. Some examples are: summer / winter, maximum feed / turndown, shut down under nitrogen blanket, maximum gasoline / maximum diesel production.
Static data	Data that represents physical equipment. Static data is desired to be the same between all source and target cases. Some examples are: valve Cv, vessel dimensions, heat exchanger area, pump curves.
Dynamic data	Data that represents operating conditions. Dynamic data is expected to be different between source and target cases. Some examples are: stream temperature, pressure and composition, controller setpoints, valve actuator position, ambient temperature.
Initialize From	The dynamic data is copied from the source case to the target case. The static data in the target case is unchanged.

When managing dynamic cases, a typical use case would be for the user to Initialize From a source case to a target case, integrate until operation is lined out, and then Save As over the source case. The result will be that the dynamic data (operating conditions) will be different between the two cases, but the static data (equipment sizes) will be the same.

7.8.2 Dynamic Usage Notes

The Initialize From operation will copy the dynamic data for every matching unit operation from the source to the target case. Therefore, the target case should contain a superset of all of the unit operations.

The Initialize From functionality matches unit operations between the source and target cases based on the name of the unit operation. Therefore, if the name of a unit operation has been changed between the source and target case, then that unit operation will not be initialized. If a unit operation is not initialized, then after the Initialize From operation completes, the dynamic data for that unit operation will be based on the data from the target case, and not the source case. All other unit operations (provided they completed successfully) will have dynamic data from the source case. When you start the integrator, there is likely to be a significant "bump" to the plant operation.

Note that if sub-flowsheets are used, the name of the unit operation includes the tag of the sub-flowsheet. Therefore, if the tag of a sub-flowsheet has been changed between the source and target case, then none of the unit operations in that sub-flowsheet will be initialized.

The dynamic data is highly dependent on the thermodynamic calculations. Therefore, if the Fluid Package used for a unit operation is changed between the source and the target, then that unit operation will not be initialized. Dynamic Initialize From is not supported for UniSim Thermo.

It is recommended that you integrate both the source and target cases and save them before starting the Initialize From operation. Firstly, many internal variables are calculated just after the green light is turned on. Secondly, if you change an equipment size, some internal variables are reset at the time of the change. They are not recalculated until the green light is turned on. Therefore, always integrate the case after making a change. If you save an integrated case, all of the internal variables will be saved with the case. However, if you Initialize From using an unintegrated case, you will get many uninformative warnings and errors.

For dynamic cases, the variable status determines a variable's colour in

the view: blue is specified, red is default, and black is calculated. The Initialize From operation will not copy the variable status from the source to the target. Consequently, a variable that was red in the source case might be blue in the target case.

If a static variable (valve Cv, heat exchanger size, pump curve, etc.) is changed, it is highly likely that there will be a bump after Initialize From (i.e. the case will not operate at steady-state). A warning message is issued for some of these changes, but not all.

Notes are considered static data, so the superset of notes should always be in the target case.

It is possible to Initialize From between two steady-state cases, or two dynamic cases, but not between a steady-state and a dynamic case.

During an Initialize From operation, the source case is loaded just like it would be if you opened the case, only it is loaded into memory without displaying the user interface. Consequently, it is recommended that your preferences are set so that the checkbox for "Confirm Before Adding if Active Correlations are Present" is not ticked (on the Simulations tab, Options page). Otherwise, you will have to confirm every time the Initialize From operation is used.

Note that the determination of whether a variable is static or dynamic is subjective. Most variables are obviously static or obviously dynamic, however, some could be either static or dynamic depending on the situation. Therefore, for some unit operations, the user is allowed to choose whether a variable will be initialized (dynamic) or not (static). The following sections describe the special behaviours and usage for each unit operation type during Initialization From for dynamic cases.

Streams

Material Streams

For each individual stream in the flowsheet:

- "if the pressure specification is active, you can choose to make it static or dynamic using the check box on the Dynamics tab, Specs page
- "if the flow specification is active, you can choose to make it static or dynamic using the check box on the Dynamics tab, Specs page

- "if the stream has a feeder block, you can choose to make the conditions (temperature and composition) static or dynamic using the check box on the Feeder block view (called up from the message box on the Dynamics tab, Specs page), Composition tab.
- "if the stream has a product block, you can choose to make the reverse flow conditions (temperature and composition) static or dynamic using the check box on the Product block view (called up from the message box on the Dynamics tab, Specs page), Composition tab.

Each stream is flashed after the Initialize From operation is completed.

Energy Streams

The heat flow is a dynamic variable.

Heat Transfer Equipment

Air Cooler

Demanded fan speed and on/off are dynamic. Design fan speed, design fan air flow, UA, and pressure drop factors are static.

Initialization will fail if the row/pass configuration, or number of fans is changed.

Initialize From for UniSim CFE is not supported.

Cooler/Heater

For all heater/coolers the sizing and configuration variables are static. For each individual heater/cooler, the temperature, duty, or duty fluid specifications can be made static or dynamic using the check box on the Dynamics tab, Specs page.

Initialization will fail if the number of zones or specification type (temperature, duty, or duty fluid) is changed.

Heat Exchanger

The sizing and configuration variables are static. UA and UA reference flow are static.

Initialization will fail if the number of zones or TEMA type is changed.

Initialize From for UniSim STE is not supported.

Fired Heater

The sizing and configuration variables are static.

LNG

The sizing and configuration variables are static. The U calculation method is static.

Initialization will fail if the number of zones or layers is changed.

Initialize From for UniSim PFE is not supported.

Piping Equipment

Mixer

The pressure specification is static. The Product Molar Flow Factor is dynamic.

Pipe Segment

The sizing and configuration variables are static. If the order of the fittings is changed, initialization will proceed, but there is likely to be a bump to the plant operation.

Initialization will fail if the number of segments is changed.

Compressible Gas Pipe

Not supported for USD dynamics - only available in steady-state.

Tee

If splits are used as flow specifications, they are dynamic.

Valve

The sizing and configuration variables are static. The valve actuator position is dynamic. Instructor failures of failed actuator and worn trim are dynamic.

Limit switches are matched between the source and target cases based on name (so the order does not matter). If a limit switch was included in the source case but not the target, or its name was changed, a warning message is displayed.

Relief Valve

The sizing and configuration variables are static. Valve failure is dynamic.

Rotating Equipment

Centrifugal Compressor or Expander

The sizing and configuration variables are static. The specification types (head or speed, etc.) are static, but the specification values are dynamic. Use of electric motor is static, but a warning message is displayed if it differs between the source and target case.

Reciprocating Compressor

The sizing and configuration variables are static. The specification types (head or speed, etc.) are static, but the specification values are dynamic. Variable clearance volumes are dynamic. Enabling variable clearance volumes, and use of electric motor are static, but a warning message is displayed if they differ between the source and target case.

Centrifugal Pump

The sizing and configuration variables are static. The specification types (head or speed, etc.) are static, but the specification values are dynamic. Use of electric motor is static, but a warning message is displayed if they differ between the source and target case.

Malfunction state and information are dynamic.

Positive Displacement Pump

The sizing and configuration variables are static. The specification types (head or speed, etc.) are static, but the specification values are dynamic. Use of electric motor is static, but a warning message is displayed if they differ between the source and target case. Malfunction state and information are dynamic.

Separation Operations

Separator, 3-Phase Separator, & Tank

The sizing and configuration variables are static.

Level taps are matched between the source and target cases based on name (so the order does not matter). If a level tap was included in the source case but not the target, or its name was changed, a warning message is displayed.

If the volume is changed between the source and target, the holdup mols are automatically adjusted. This will likely prevent a bump if there is only vapour holdup. If there is liquid holdup, it is unlikely that the level controller will be at steady-state.

Shortcut Column

Not supported for USD dynamics - only available in steady-state.

Component Splitter

For all component splitters, the temperature and pressure specifications are static. For each individual component splitter, the component or TBP split specifications can be made static or dynamic using the check box on the Design tab, Splits or TBP Cut Point page.

Column

The sizing and configuration variables (including tray efficiencies and pressure drop factors) are static.

If the tray volume is changed (diameter or tray spacing) between the

source and target, the holdup mols are automatically adjusted; however, there is still likely to be a bump to operation.

Initialization will fail if the number of trays is changed.

Reactors

Conversion Reactor, Equilibrium Reactor, CSTR

The reaction sets, reactions, and kinetic constants are all static. The sizing and configuration variables are static.

Initialization will fail if the number of reaction sets or reactions is changed.

If the volume is changed between the source and target, the holdup mols are automatically adjusted. However, this is likely to affect the reaction rates, so a bump to operation is expected.

Gibbs Reactor

The Gibbs Reactor is not recommended for use in dynamics.

Plug Flow Reactor

The reaction sets, reactions and kinetic constants are all static. The sizing and configuration variables are static.

Initialization will fail if the number of reaction sets, reactions, or segments is changed.

If the volume is changed between the source and target, the holdup mols are automatically adjusted. However, this is likely to affect the reaction rates, so a bump to operation is expected.

Solid Separation Operations

Simple Solid Separator, Rotary Vacuum Filter, Baghouse Filter, Centrifuge

Not supported for USD dynamics - only available in steady-state.

Screen, Cyclone, Hydrocyclone, Crusher

Initialize From supported for USD dynamics.

Conveyor

The sizing and configuration variables (including belt/bucket, acceleration speed, length, capacity, etc.) are static. The specified speed is dynamic.

Boolean Operations

On/Off Delay Gate

Delay Time is static.

If a source case was saved where the input had changed, but the output had not yet changed because the delay time had not been reached, the output will change based on the cumulative integration time since the input was changed. For example, if the delay time is 10 minutes, and a case is saved 5 minutes after the input changes, the output will change 5 minutes after the Initialize From operation.

Counter Up/Down Gate

Maximum counter, PV alarm, and Desired Output value are static.

Cause and Effect Matrix

The matrix configuration variables (including trip points, delay times, and descriptions) are static. Input (causes) bypasses and Output (effects) overrides are dynamic. Whether an Output Local Switch exists is static; the switch value is dynamic.

Inputs and outputs are matched between the source and target cases based on name. If a cause or effect was included in the source case but not the target, or its name was changed, a warning message is displayed.

Control Operations

Split Range Controller, Ratio Controller, PID Controller

The sizing and configuration variables (including controller action, tuning constants, and pv/sp/op range) are static. Mode, set point, and output are dynamic. Execution (internal or external) is static, but a warning message is displayed if they differ between the source and target case.

For Set Point Ramping, Ramp Duration is static, but Ramp Enable/Disable and Target SP are dynamic. If a source case was saved where the ramp had not completed, the ramp will continue at the same slope after an Initialize From operation.

Autotuning is not supported by Initialize From for USD dynamics.

The malfunction aspect of PV Conditioning is dynamic.

MPC Controller, DMCplus Controller, Profit Controller

Initialize From not supported for USD dynamics.

Control Valve

The Valve Sizing data (flow type, min/max flow) are all static.

Control OP Port

The Control OP Port data (min/max value) are all static.

Selector Block

The sizing and configuration variables (including mode, and gain/bias) are static.

Transfer Function

The sizing and configuration variables (including active transfer functions) are static.

Digital Point

The sizing and configuration variables (including threshold and dead band) are static. Mode and output are dynamic.

Spreadsheet

For all spreadsheets, the formulae are static. For each individual spreadsheet, each user entered constant can be made static or dynamic using the check box on the Initialize From tab.

Spreadsheets have special handling for Initialize From because they can operate on values imported from other unit operations and can export calculated values to other unit operations. Consequently, all spreadsheets are calculated after the Initialize From operation has updated the dynamic data for all other unit operations. This eliminates possible calculation order dependencies once the integrator is started.

Integrator

Integration step size, and options for static head, heat loss, rigorous non-equilibrium mixed properties, liquid valve choking, and reduced liquid efficiency for low vapour holdups are static, but a warning message is displayed if they differ between the source and target case. Default ambient temperature and the Current Time are dynamic.

Strip Charts

The Sample Interval is static. The historical data is dynamic. The Logger Size (number of samples) is taken as the maximum between the source and target case. If the Logger Size or Sample Interval is different between the source and target cases, all past data will be based on the Logger Size and Sample Interval of the source case; all future data will be based on the Sample Interval of the target case.

Strip Charts and Individual Strip Chart Data Selections are matched between the source and target cases based on name. If a Strip Chart or Data Selection was included in the source case but not the target, or

its name was changed, a warning message is displayed.

Extensions

Extension variables are defined in the edf. All variables with the attribute TriggerSolve are dynamic; variables with NoTriggerSolve are static.

Steady-State Operations

The following steady-state operations are not supported by Initialize From for dynamic cases:

- Adjust
- Set
- Recycle
- Balance

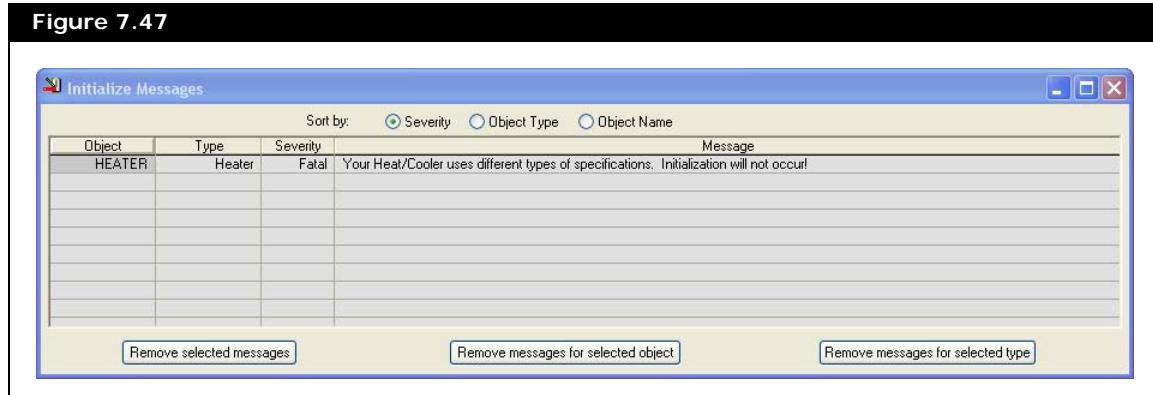
7.8.3 Steady-State Usage Notes

Initialize From functionality for steady-state cases will be included in a future release.

7.8.4 Warning Messages

After the Initialize From operation has completed, an analysis of the changes are displayed. Note that if there were no error/warning messages, this view will not be displayed.

Figure 7.47



Sort by radio buttons	Use these radio buttons to sort the Messages by either: Severity, Object Type, or Object Name.
Message matrix	Displays the messages. If you double-click on an object name, the view for that unit operation will be displayed.
Remove selected messages	Single or multiple messages can be selected and removed
Remove messages for selected object	Messages for single or multiple objects can be selected and removed
Remove messages for selected type	Messages for single or multiple object types can be selected and removed

Following the Initialize From operation, if the model is not running at steady-state, the analysis display can be used to help trace the root of the problem. The message severity levels have the following meaning:

Fatal	The unit operation could not be initialized. This could be because: the unit operation did not exist in the source case, it used a different fluid package, or there was a significant change to the structure of the unit operation (such as number of zones in a heat exchanger), etc.
Critical	Only a partial initialization of the object occurred. This could be because: one separator models carry over and the other doesn't, one compressor has a rotational linker and the other doesn't, etc.
Warning	Initialization completed, but some change to the data or structure of the unit operation is likely to cause a bump to operation.

7.9 Dynamic/Steady State Modes

Press the **F7** hot key to alternate between the two modes.

The Dynamic/Steady State command is located in the Simulation menu and is used to alternate between the two modes.

To switch from Steady State to Dynamic mode, open the Simulation menu and select the Dynamics command. The Dynamics Assistant identifies items that require attention:

- Click **Yes** to open the Dynamics Assistant and view the items.
- Click **No** to ignore them and continue to Dynamics mode.

To switch from Dynamic to Steady State mode, open the Simulation menu and select the Steady State command. Switching back to Steady State mode results in the loss of results:

- Click **Yes** to proceed to steady state.
- Click **No** to remain in dynamics.

See [Chapter 2 - Dynamic Tools](#) in the **UniSim Design Dynamic Modeling Guide** for additional information about the Dynamics Assistant.

Your steady state simulation remains unsolved and may require some adjusting before it fully solves.

7.10 Solver Active/Holding

When inheriting a case from someone else, double-check that the solver is active when your case is not solving.

When in Steady State mode, you can activate or hold the solver. This is useful for building your simulation without interruption; you can add operations and streams without UniSim Design trying to solve them immediately. By default the UniSim Design solver is active.

Use **one** of the following methods to place the solver on hold:

- Open the Simulation menu and click the **Solver Holding** command.
- Press the **F8** hot key.
- Click the **Solver Holding** icon in the toolbar.



Solver Holding icon

Use **one** of the following methods to make the solver active:

- Open the Simulation menu and click the **Solver Active** command.
- Press the **F8** hot key.
- Click the **Solver Active** icon in the toolbar.



Solver Active icon

7.11 Integrator Active/ Holding

When in Dynamics mode, you can activate and hold the integrator. This is useful for examining the results of your simulation. By default the integrator is on hold (inactive).

Use **one** of the following methods to activate the integrator:

- Open the Simulation menu and click the **Integrator Active** command.
- Press the **F9** hot key.
- Click the **Integrator Active** icon in the toolbar.



Integrator Active icon

Use **one** of the following methods to put the integrator on hold:

- Open the Simulation menu and click the **Integrator Holding** command.
- Press the **F9** hot key.
- Click the **Integrator Holding** icon in the toolbar.



Integrator Holding icon

7.12 Equation Summary

The Equation Summary view is used when running a case in Dynamics mode and it automatically opens when there are dynamic specification errors in your case.

The Equation Summary option is available only in Dynamics mode.

Refer to [Section 2.3 - Equation Summary View](#) in the **UniSim Design Dynamic Modeling Guide** for additional information.

To open the view manually, select the Equation Summary View command from the Simulation menu.

The Equation Summary provides a list of the equations and pressure flow specifications that are currently used in the dynamic simulation. It also enables you to analyse the simulation to determine if any equations/specifications are required or redundant.

7.13 Enter Basis Environment

Use the hot key **CTRL B** to enter the Basis environment from any environment.



Enter Basis Environment icon

Refer to [Chapter 5 - Basis Environment](#) of this guide and the **UniSim Design Simulation Basis Guide** for additional information about the Basis environment.

To enter the Basis environment from any flowsheet, use one of the following methods:

- Click the **Enter Basis Environment** command in the Simulation menu.
- Click the **Enter Basis Environment** icon in the toolbar.

The Basis environment allows you to create and manipulate the fluid package in your simulation. Each object and flowsheet can have its own fluid package, and flowsheets can contain multiple fluid packages. For each fluid package, you can supply the following:

- Property package
- Components
- Reactions
- User properties

7.14 User Variables

Refer to [Chapter 5 - User Variables](#) in the **UniSim Design Customization Guide** for additional information.

User Variables increase the internal functionality of objects, such as streams and unit operations, by dynamically attaching variables and code to those objects from within the application.

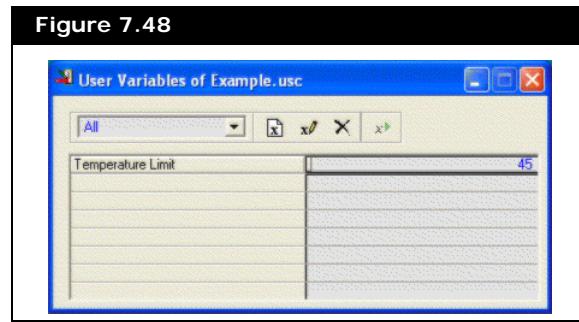
User Variables cannot be distinguished from variables built into UniSim Design objects so they can be added to spreadsheets, targeted by logic controllers, and have their values specified by user input.

For example, you can attach a User Variable to a stream to ensure the flow rate is specified lower than a certain value, or display a view when a vessel temperature exceeds a certain value.

User Variables let you attach code (written in a Visual Basic compatible macro language) to simulation objects and specify when the code is to execute. This can add extra functionality to any simulation.

Simulation Case User Variables

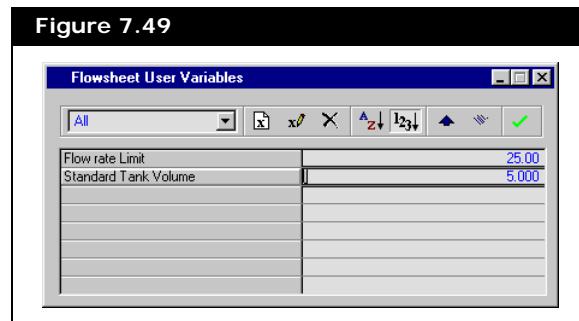
Select the Simulation Case User Variable command from the Simulation menu to access the User Variables of Case view.



The User Variables listed in this view apply to the current simulation case.

Flowsheet User Variables

Select the Flowsheet User Variables command from the Flowsheet menu to access the Flowsheet User Variables view.



The User Variables listed in this view apply only to the flowsheet.

7.14.1 Adding a User Variable

You can add a User Variable in either the User Variables of Case view or the Flowsheet User Variables view.



Create a New User Variable icon

1. Click the **Create a New User Variable** icon to add a new User Variable to your simulation. The Create New User Variable view appears.
2. On the Create New User Variable view, you can enter code to define your User Variable. You can also specify the following:
 - Type
 - Dimensions
 - Units
 - Macros
 - Attributes
 - Filters
 - Security
 - Variable defaults

7.14.2 Editing a User Variable

You can edit a User Variable from either the User Variables of Case view or Flowsheet User Variables view.



Edit the Selected User Variable icon

1. Select the User Variable you want to edit from the list of available User Variables.
2. Click the **Edit the Selected User Variable** icon. The Edit Existing Code of Case User Variables view appears. This view allows you to modify the code, tag, type and dimensions of the user variable.

7.14.3 Deleting a User Variable

You can delete a user variable from either the User Variables of Case or Flowsheet User Variables view.



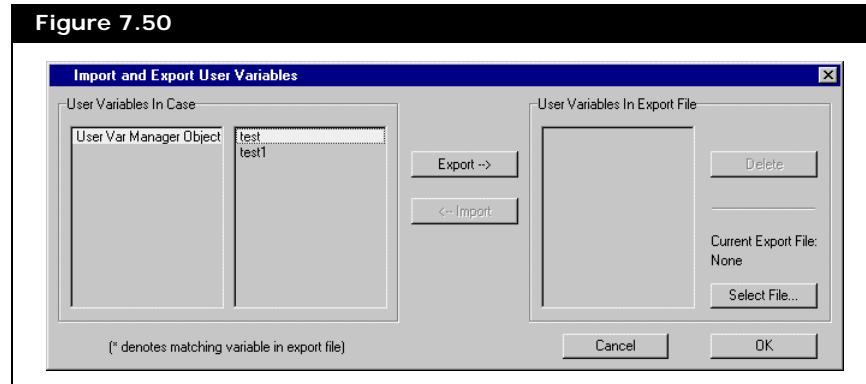
Delete the Selected User Variable icon

1. Select the User Variable you want to delete from the list of available User Variables.
2. Click the **Delete the Selected User Variable** icon. You are prompted to verify the deletion of the User Variable.

7.15 Importing & Exporting User Variables

From the Simulation menu, select the Import and Export User

Variables. The Import and Export User Variables view appears.



7.15.1 Importing User Variables

1. Click the **Select File** button. The Open File view appears.
2. Browse to the location of the User Variable file you want to import (*.hvv).
3. Select the file you want to import and click **Open**.
4. From the list of available User Variables in the export file, select the User Variable(s) you want to import.
5. Click the **Import** button.

Select multiple User Variables at one time by holding down the **CTRL** key, and clicking each variable you want to select.

Select multiple User Variables at one time by holding down the **CTRL** key and clicking each variable you want to select.

7.15.2 Exporting User Variables

1. Click the **User Var Manager Object** in the User Variables In Case group. A list of available User Variables appears.
2. From the list of available User Variables, select the User Variable(s) you want to export.
3. Click the **Export** button. The Save File view appears.
4. Specify a name and location for the User Variable file and click **Save**.

7.16 Oil Output Settings

Select the Oil Output Settings command in the Simulation menu to open the Oil Output Settings view. For more information, see [Section 6.2.1 - Oil Output Settings View](#).

7.17 Object Navigator

The Object Navigator is one of the two navigational aids that brings the multi-flowsheet architecture into a flat space. A simulation containing a main flowsheet and sub-flowsheets (columns and/or template sub-flowsheets) is 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.

One difference, however, is that sub-flowsheets are both flowsheet elements (within the main simulation) as well as flowsheets themselves. How you use the navigational tools is illustrated in the following subsections.

The Object Navigator enables you to locate and view any flowsheet element within any flowsheet, or enter the build environment for a flowsheet.

There are several ways to access the Object Navigator:



Object Navigator icon

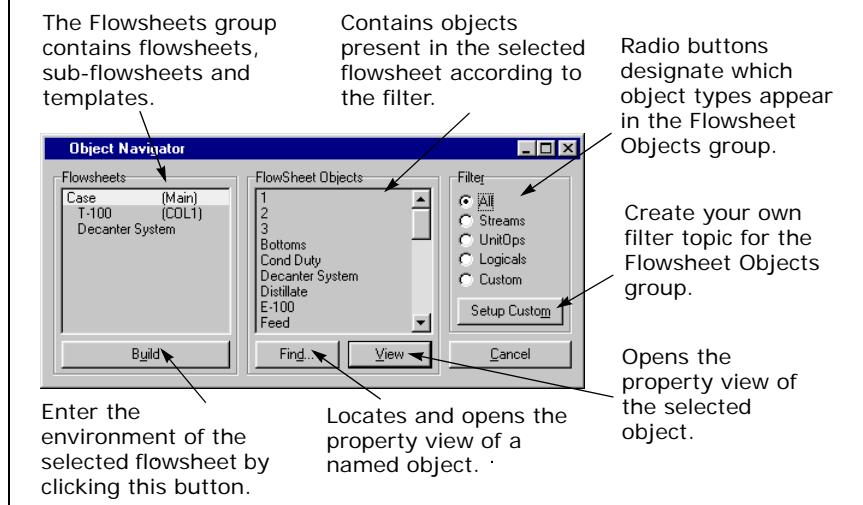


Object Inspect menu

- Click the **Object Navigator** icon in the toolbar.
- Click the **Find Object** command in the **Flowsheet** menu.
- Press the **F3** hot key.
- Right-click in any blank area of the UniSim Design Desktop and select the **Find Object** command from the Object Inspect menu.

The Object Navigator view appears.

Figure 7.51



Object Filter Tools

Filter Radio Buttons

These radio buttons allow you to filter the Object list by certain types. For example, click the Streams radio button to view only streams.

Setup Custom Button

Click the '+' symbol to expand the tree and view more options.

The Setup Custom button lets you to define your own filter criteria.

1. Click the **Custom** radio button or the **Setup Custom** button. The Select Type view appears.
2. Select the stream type or unit operation type.
3. Click **OK**.

Entering the Build Environment

Enter the Build environment for any flowsheet directly from the Object Navigator.

1. From the list of available flowsheets, select the flowsheet you want to enter.
2. Click the **Build** button.

7.17.1 Locating an Object

The Object Navigator works in a left to right sequence. When you select a flowsheet, all flowsheet objects (based on the current filter) appear in the Flowsheet Objects group.

Use this procedure to locate an object and open that object's property view.

1. From the list of available flowsheets, select the required flowsheet.
2. In the Flowsheet Objects list, select the object you want to view.
3. Click the **View** button. The Object Navigator is closed and the property view for the selected object appears.

Use this procedure to locate an object by name.

1. Click the **Find** button. The Find Object view appears.
2. Type the name of the object in the **Object Name** field.
3. Click **OK**. The Object Navigator is closed and the property view for the selected object appears.

Use any one of the following methods to access a property view for a specific flowsheet object. For these methods you need to first select the appropriate flowsheet from the Flowsheets group.

- Select the required object in the **Flowsheet Objects** group, then click the **View** button.
- Double-click the object in the **Flowsheet Objects** group.
- Click the **Find** button, then type the name of the object you want to locate.

You can start or end the search string with an asterisk (*), which acts as a wildcard character. This lets you find multiple objects with one search. For example, searching for VLV* will open the property view for all objects with VLV at the beginning of their name.

When the required object is located, the Object Navigator closes and the property view for that stream or operation is opened.

7.18 Simulation Navigator

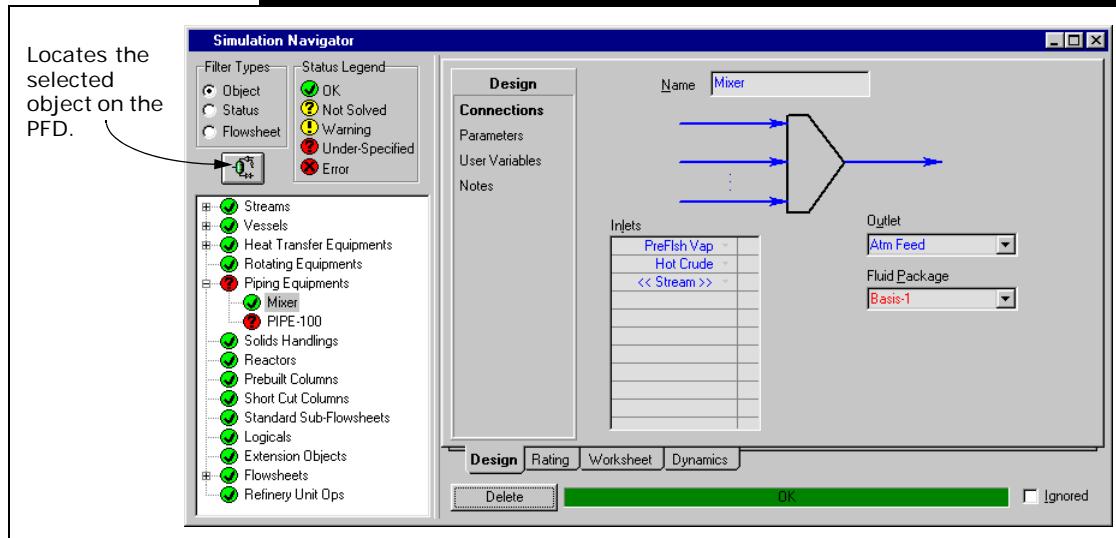
The Simulation Navigator lets you quickly access the property view of any stream or unit operation within the case. Within the Simulation Navigator view, you can modify object parameters the same way you would in the regular object property view.



Simulation Navigator icon

To access the Simulation Navigator, select the Simulation Navigator command from the Flowsheet menu, or click the Simulation Navigator icon in the toolbar.

Figure 7.52



In the Filter Types group, use the radio buttons to filter the flowsheet objects:

- **Object Filter.** Organizes the attachment tree by object type. This allows you to access all objects of a given type (i.e., heat exchangers or vessels) regardless of the flowsheet they exist in.
- **Status Filter.** View all of the objects by status type: OK, Not Solved, Warning, Under Specified and Error.
- **Flowsheet Filter.** All the objects within a column or sub-flowsheet appear under their respective column/sub-flowsheet title. The objects in the main flowsheet are listed in alphabetical order.

For certain objects you may need to enlarge the view horizontally to see more of the property view. To resize the view, place the mouse pointer at the view edge, then click and drag the mouse button to resize the view.

7.18.1 Viewing an Object

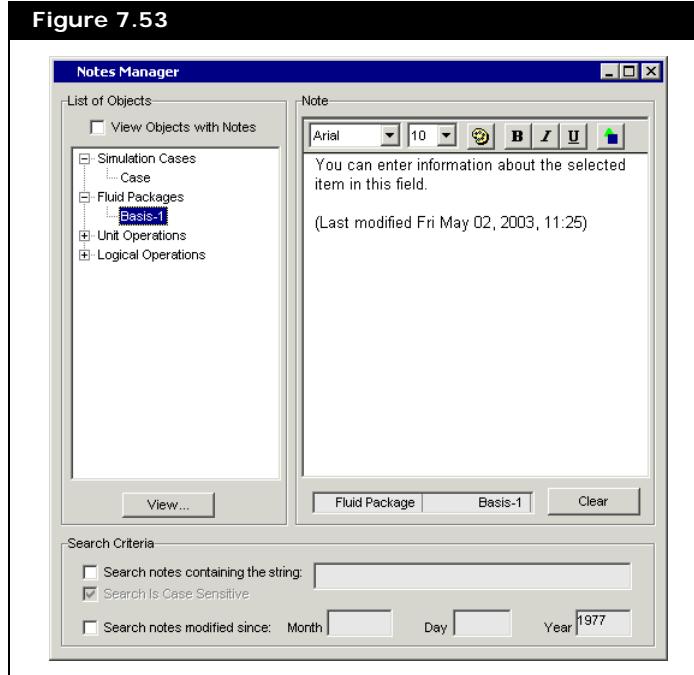
To view an object using the Simulation Navigator, do the following.

1. In the Filter Types group, select the required filter radio button.
2. From the list of available objects, select the object you want to view. Click the '+' symbol to expand the tree and reveal more selections.

You can use the Object Inspect menu (right-click) to access commands associated with the current property view.

7.19 Notes Manager

The Notes Manager lets you search for and manage notes for a case.



To access the Notes Manager, select the Notes Manager command from the Flowsheet menu, or press the **CTRL G** hot key.

View/Add/Edit Notes

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

Click the icon to expand the tree browser.

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

Search Notes

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

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

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

The case sensitive search option is only available if you are searching by string.

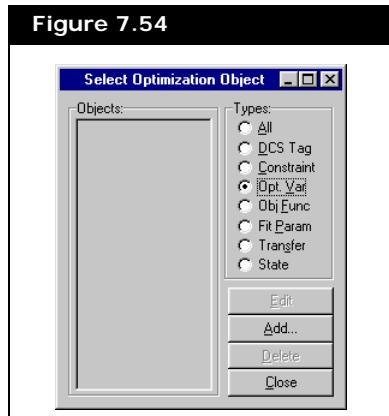
- Check the **Search notes modified since** checkbox, then type a date. Only objects with notes modified after this date will appear in the object list.

7.20 Optimization Objects

Refer to **Chapter 2 - Using UniSim Design.RTO** in the **UniSim Design RTO Reference Guide** for additional information.

The Optimization Objects view lets you select a generic set of objects that identify the underlying flowsheet variable(s) and provide the necessary configuration information for use by Optim or Estim.

To access the Select Optimization Object view, select the Optimization Objects command from the Flowsheet menu.



7.20.1 Adding an Optimization Object

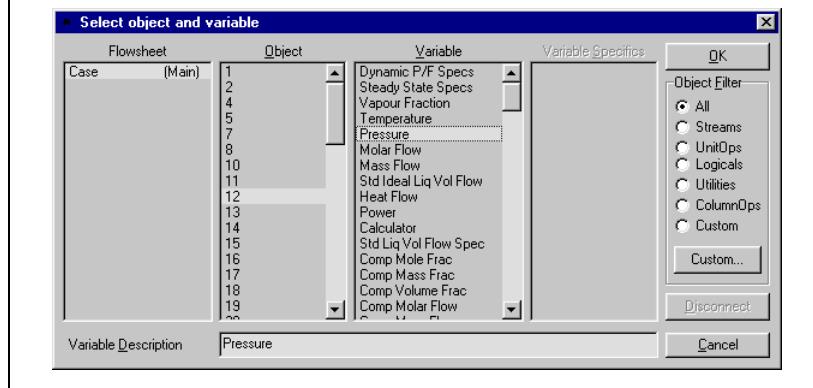
1. In the Types group, select one of the optimization type radio buttons to determine the type of optimization object you are creating.

You cannot add an optimization object when the All radio button is selected.

2. Click the **Add** button.

The Select object and variable view appears.

Figure 7.55



You must select an object before the variable list is populated.

3. Select the object and variable you want to optimize.
4. Click **OK**. The Optimization Object view appears.
5. Specify the parameters defining your optimization object.

7.20.2 Editing an Optimization Object

1. In the Select Optimization Object view, select the object you want to edit from the Objects list.
2. Click the **Edit** button. The Optimization Object view appears.
3. Modify the parameters that define your optimization object.

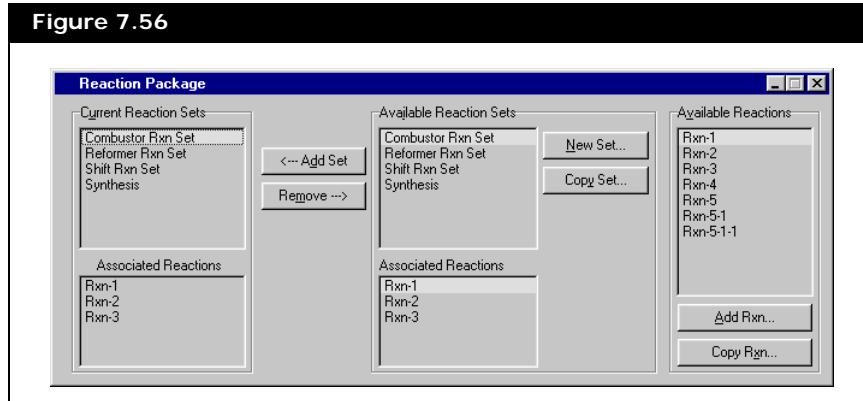
7.20.3 Deleting an Optimization Object

1. Select the object you want to delete from the list of available objects.
2. Click the **Delete** button. UniSim Design prompts you to confirm the deletion.

7.21 Reaction Package

Refer to [Chapter 5 - Basis Environment](#) in this guide and [Chapter 5 - Reactions](#) in the **UniSim Design Simulation Basis Guide** for more information about reactions and reaction sets.

To access the Reaction Package view, select the Reaction Package command from the Flowsheet menu.



This view allows you to 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 in the current case

The Reaction Package view eliminates the need to return to the Reaction tab of the Simulation Basis Manager to define reactions and reaction sets. The only aspect of defining reactions that must be done in the Simulation Basis Manager is the selection of components.

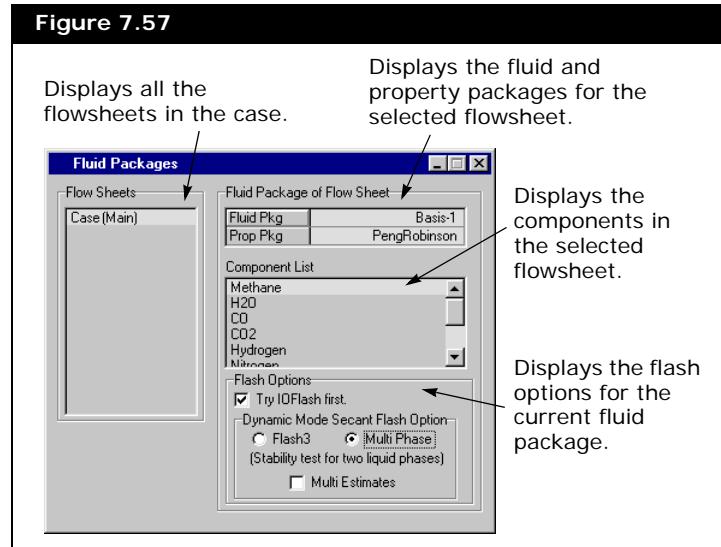
7.22 Fluid Package/Dynamics Model

To access the Fluid Packages view, select Fluid Package/Dynamics

For information regarding the Dynamic Flash options, refer to **Section 2.4.4 - Stability Test Tab** of the **UniSim Design Simulation Basis Guide**.

Model from the Flowsheet menu.

The Fluid Packages view provides you with a summary of all the fluid packages in the simulation and lets you 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.



If flowsheets use the same fluid package, they must also use the same flash options.

7.23 Workbook

The most concise way to display process information is in a tabular format. The Workbook is designed for this purpose and extends the concept to the entire simulation. In addition to displaying stream and general unit operation information, the Workbook is also configured to display information about any object type (streams, pipes, controllers, separators, etc.).

The Workbook becomes a collection of tabs. For example, if you add a tab for Separators, then every separator in the flowsheets appear on the tab with their current value of process variables displayed. To provide the greatest degree of flexibility, modify the variable set to show the variables of interest, or install multiple tabs for the same object type with varying levels of detail.

Not only is the Workbook useful for process analysis, but it was also developed as an integral element in the building and manipulation of

Most of the object properties in UniSim Design are calculated based on information entered by the user, so if there are large quantities of data in your workbook, then the performance of your simulation may be affected.

For example, extensive calculations such as Heat of Comb will take longer to calculate than simpler calculations.

your simulation. In addition to displaying the process information, you can make changes to specifications directly from the Workbook and calculations are performed automatically. Mechanisms were also built into the Workbook to provide immediate access to the property view for an individual stream or operation.

Key Workbook features:

- Workbook tabs can be added/deleted as required.
- Multiple tabs for a given object type are supported, allowing each tab to display different variables for that object type.
- Objects on a given Workbook tab can be sorted, hidden, or revealed as required.
- User defined configurations of variables for given object types can be stored independently of the case they were configured in, and read back in to any other simulation case.

The Workbook can be exported entirely or as individual tabs. When importing, the user defined configurations can either replace or be appended to the Workbook.

Each flowsheet in your simulation (main flowsheet and column/template sub-flowsheets) has its own Workbook. You can access the Workbook for any flowsheet from any location in your simulation.

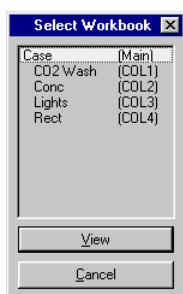
7.23.1 Opening a Workbook

There are three ways to open a Workbook:

- Click the **Workbook** icon in the toolbar.
- Select the **Workbooks** command in the **Tools** menu.
- Press the **CTRL W** hot key to open the Workbook for the current flowsheet.



Workbook icon



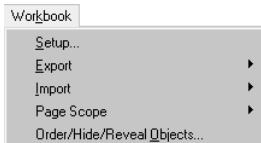
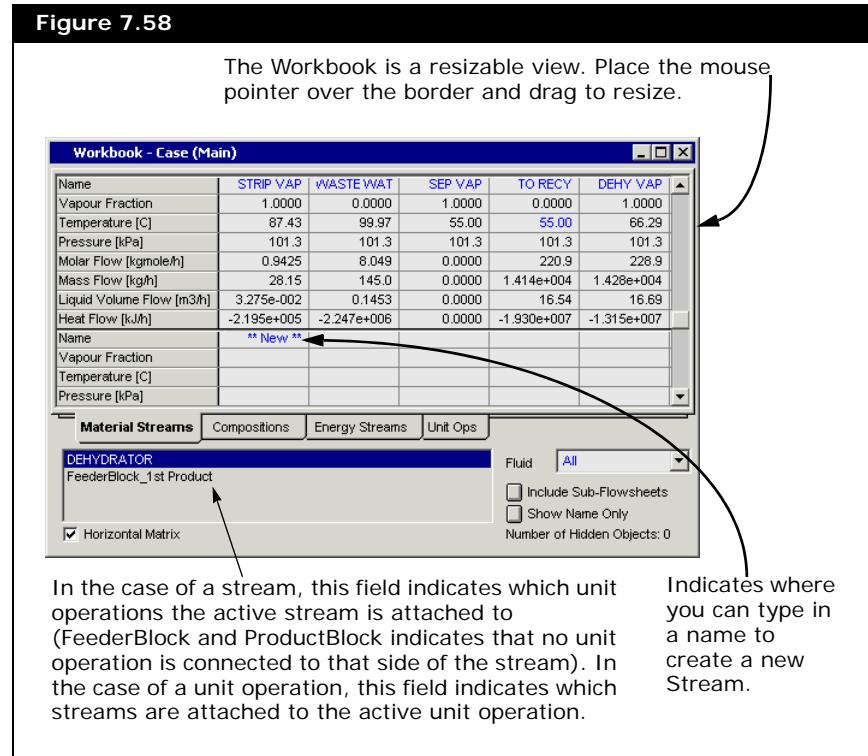
Select Workbook view

The Select Workbook view displays all flowsheets in the simulation. To open a specific Workbook, select the flowsheet containing the Workbook and click the View button.

The first time you access the Workbook, it opens to the Material Streams tab, displaying the basic stream information for all streams currently installed in the main flowsheet. The default Workbook also

contains tabs for Compositions, Energy Streams, and Unit Ops.

Figure 7.58



When the Workbook is active, the Workbook menu appears in the menu bar. The commands associated with this menu are described below:

Command	Description
Setup	Opens the Setup view allowing you to manage the tabs in your Workbook. Refer to Section 7.23.5 - Managing Workbook Tabs for more information.
Export	Accesses a submenu containing the Workbook and Pages commands. These commands let you save a Workbook or page setup. Refer to Section 7.23.7 - Exporting/Importing Workbook Tabs for more information.
Import	Accesses a submenu containing the Workbook and Pages commands. These commands replace the current Workbook or page setup with a saved setup. Refer to Section 7.23.7 - Exporting/Importing Workbook Tabs for more information.
Page Scope	Accesses a submenu containing the Show/Hide Sub-Flowsheet Objects commands. This toggle function either shows or hides sub-flowsheet objects on the active Workbook tab. Refer to the Viewing Sub-Flowsheet Objects section for more information.
Order/Hide/Reveal Objects	Opens the Order/Hide/Reveal Objects view allowing you to sort the Workbook objects either alphabetically or manually, hide Workbook objects and reveal Workbook objects. Refer to Section 7.23.6 - Sorting Information for more information.

7.23.2 Installing Streams or Operations

Streams

Use this procedure to install a new stream using the Workbook.

1. Click the **Material Streams** tab.
2. Click the ****New**** cell and type the stream name.
3. Press **ENTER** or click any other cell to complete your input.
4. Specify values for three of the properties in the table with at least one of the specifications being temperature or pressure.
5. Specify the composition of the stream.

Operations

Use this procedure to install a new unit operation through the Workbook:

1. Click the **Unit Ops** tab.
2. Click the **Add UnitOp** button. The UnitOps view appears.
3. From the list of available unit operations, click the operation you want to install.
4. Click **Add**. The operation is added to the Workbook and the operation's property view automatically opens.

7.23.3 Deleting Streams or Operations

Streams

To delete a stream from the Workbook:

- Right-click any cell associated with the stream you want to delete. From the Object Inspect menu, select the **Delete** command.
- Click the Name cell associated with the stream you want to delete and press the **DELETE** key on the keyboard.

A confirmation message appears to ensure the deletion is intended.

If the unit operation category is known, selecting the corresponding radio button in the UnitOps view filters the list of available unit operations. For example, click the Heat Transfer Equipment radio button to display only unit operations associated with heat transfer.



To delete objects without confirmation, clear the Confirm Delete checkbox on the Simulation page of the Session Preferences view. Access the Session Preferences view by selecting the Preferences command from the Tools menu.

Operations

To delete a unit operation from the Workbook:

- Click any cell associated with the unit operation you want to delete and click the **Delete UnitOp** button.
- Click the name cell associated with the unit operation you want to delete and press the **DELETE** key on the keyboard.

UniSim Design prompts you to confirm the deletion.

7.23.4 Accessing Streams or Operations

You can 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 and Compositions 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 7.59

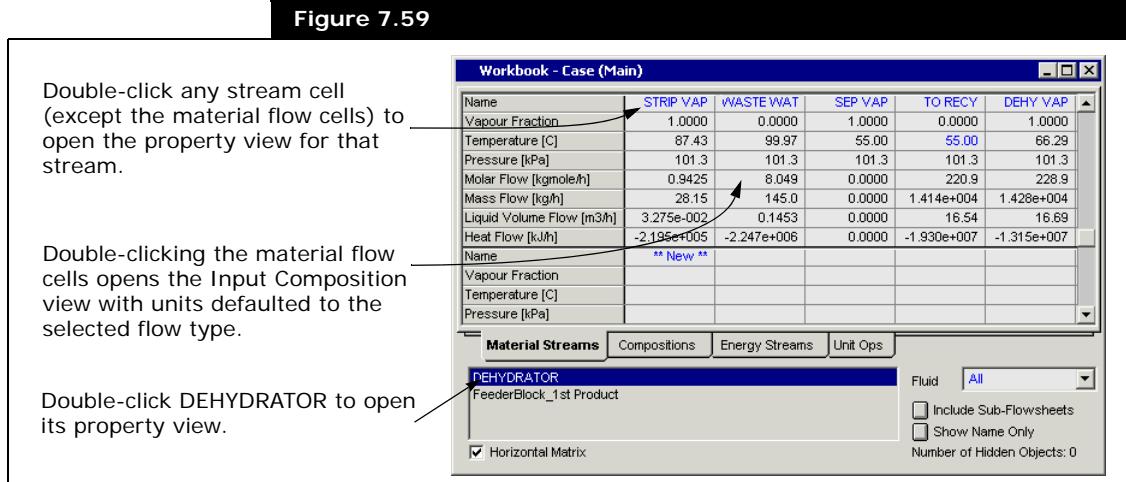


Figure 7.60

Double-click or start typing in a composition cell to open the Input Composition view for the stream.

Double-click the unit operation name to access its property view.

Double-click the name cell to access the stream property view.

Right-click any cell and select the View command to display the stream property view.

Name	1st Product	DEHY FEED	ANHYDROUS	LIGHT 1	HEAVY
Comp Mole Frac (Ethanol)	0.8831	0.4202	1.0000	0.4200	0.0425
Comp Mole Frac (H ₂ O)	0.1169	0.0109	0.0000	0.0084	0.9566
Comp Mole Frac (Benzene)	0.0000	0.5689	0.0000	0.5716	0.0009
Name	STRIP VAP	WASTE WAT	SEP VAP	TO RECY	DEHY VAP
Comp Mole Frac (Ethanol)	0.4047	0.0001	0.4044	0.4199	0.4052
Comp Mole Frac (H ₂ O)	0.5871	0.9999	0.0218	0.0109	0.0456
Comp Mole Frac (Benzene)	0.0082	0.0000	0.5738	0.5692	0.5492
Name	** New **				
Comp Mole Frac (Ethanol)					

Material Streams Compositions Energy Streams Unit Ops

DEHYDRATOR FeederBlock_1st Product Fluid All

Include Sub-Flowsheets Show Name Only Number of Hidden Objects: 0

Horizontal Matrix

Access the energy stream property views from the Energy Streams tab. The property views for unit operations that the energy streams are attached to can also be accessed through this tab.

Figure 7.61

Double-click a cell to open the energy stream property view.

Right-click a cell and select the View command to access a property view.

Double-click on the unit operation name to access its property view.

Name	DEHY REB	STRIP Q	SEP Q
Heat Flow [kJ/h]	8.416e+006	8.967e+004	8.859e+005
Name	** New **		
Heat Flow [kJ/h]			

Material Streams Compositions Energy Streams Unit Ops

DEHYDRATOR Fluid All

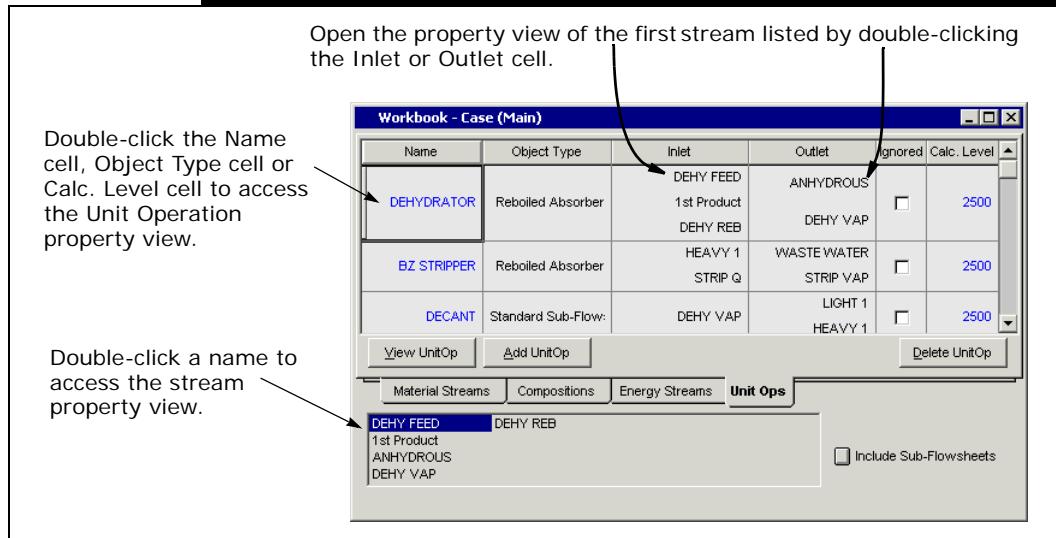
Include Sub-Flowsheets Show Name Only Number of Hidden Objects: 0

Horizontal Matrix

The same capabilities are found on the Unit Ops tab. You can access each operation's property view by double-clicking in the appropriate row. The property view of any stream that is attached to a unit

operation can also be opened from this tab.

Figure 7.62



Viewing Sub-Flowsheet Objects

From the Workbook in the main environment, you can view information for sub-flowsheet items. You can display sub-flowsheet information by clicking the Include Sub-Flowsheets button, located in the lower right corner of the Workbook. This includes all of the information from the sub-flowsheet with the information from the main flowsheet.

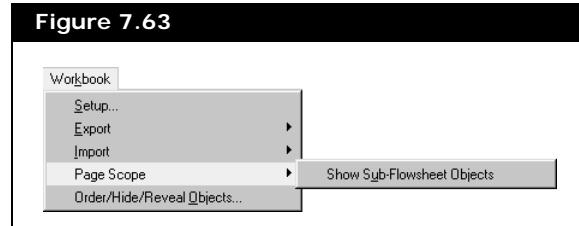
The command to display sub-flowsheet objects must be repeated on each individual Workbook tab.

The functionality of the Include Sub-Flowsheets button is not global to the entire Workbook. With each tab, you can include the sub-flowsheet objects.

To hide the sub-flowsheet objects from a Workbook tab, click the Include Sub-Flowsheets button again.

You can also view or hide sub-flowsheet items on a Workbook tab by using the Workbook menu. Under Workbook menu, click Page Scope. From the submenu, click Show/Hide Sub-Flowsheet Objects as shown below.

Figure 7.63



When the Include Sub-Flowsheets option is checked, the Composition tab only displays 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.

Show Name Only

The Show Name Only button is not available on the Unit Ops tab.

To simplify the search for a particular stream or unit operation, click the Show Name Only button, located in the lower right corner of the Workbook.

This button hides all object data except the object name. Place the cursor on the name of the chosen object and click the Show Name Only button again. All object data re-appears and the cursor remains on the selected object.

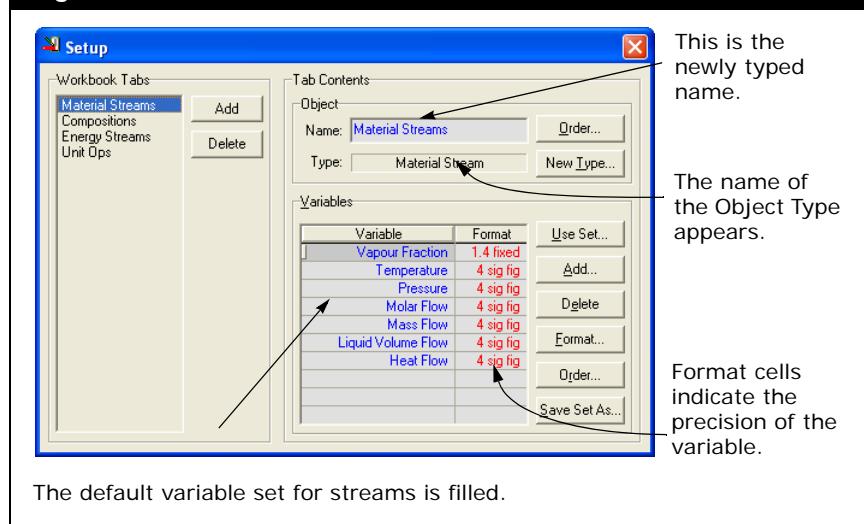
The functionality of the Show Name Only button is not global to the entire Workbook. With each Workbook tab, you can show either just the object name or all object data.

7.23.5 Managing Workbook Tabs

The Workbook Setup view lets you add, delete, and customize the tabs in the Workbook. To access this view, do the following:

- Select the **Setup** command from the **Workbook** menu.
- Right-click any of the tabs available in the Workbook. From the Object Inspect menu click the **Setup** command.

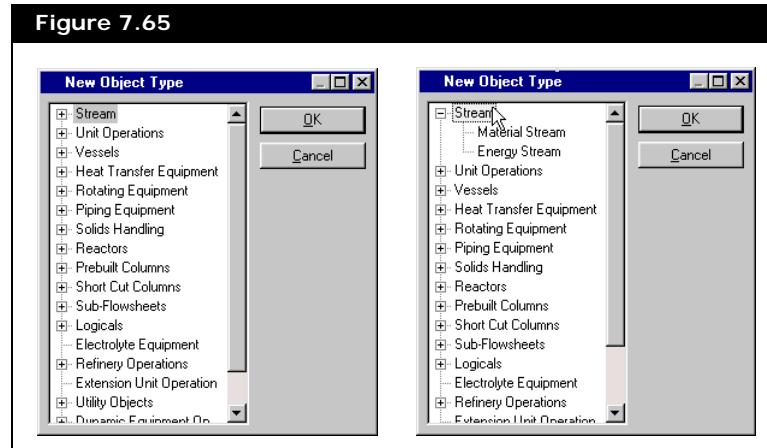
Figure 7.64



Adding a New Tab to the Workbook

1. From the list of available Workbook tabs, click the tab you want to insert the new tab before.
2. Click the **Add** button in the **Workbook Tabs** group to display the New Object Type view.
3. Click the object that is the subject of the tab. Click the **Expand** icon  to view the additional sub-items.
4. Click **Expand** icon  next to the **Stream** object type. The list appears as shown in the figure below (on the right).

Figure 7.65



The Material and Energy Stream options appear in the list.

5. Click the **Material Stream** object type and then click the **OK** button. You return to the Setup view and the new tab appears in the list of available tabs.
6. The tab is named **Material Streams 1**, since there is already a Material Streams tab. To change the tab name, click the **Name** field and type a new name for the tab.
7. Click the **Close** icon to return to the Workbook. The newly added tab is active.

Editing a Workbook Tab

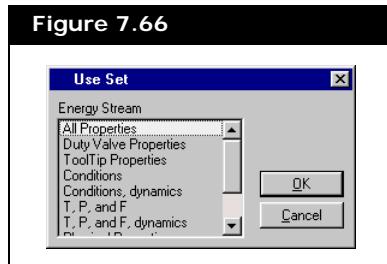
Changes cannot be made to the default Unit Ops Workbook tab.

Refer to [Section 7.23.6 - Sorting Information](#) for details on sorting objects and variables.

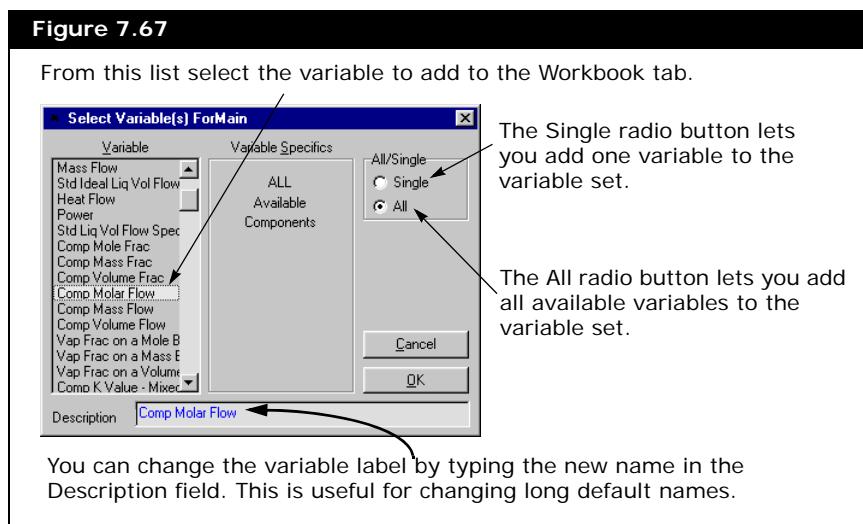
Use the Setup view to edit the tabs in your Workbook by adding, deleting, and sorting variables on the Workbook tab. Tabs can contain default variable sets or user-defined variable sets. You can define the format of each variable and modify the name, type and order of tab objects.

1. From the list of available Workbook tabs, select the tab you want to edit.
2. Click the **New Type** button to change the object type of the tab. The New Object Type view appears.

3. From the list of available objects, select the new object type, then click **OK**.
4. Click the **Use Set** button to display the Use Set view. This view lets you select a pre-defined variable set.



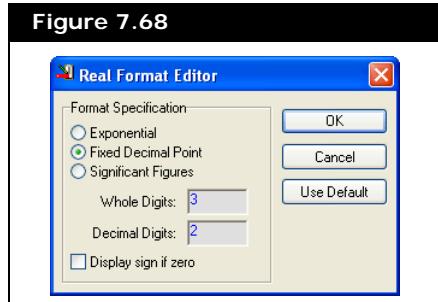
5. From the list of available variable sets, select the variable set being used, then click **OK**. The Use Set view closes and the variables appear in the Variables table.
6. To add a variable to the variable set, click the **Add** button in the Variables group. The Select Variables view appears.
7. Select the variable and any variable specifics you want to add to the variable set, then click **OK**.



Refer to [Section 10.5 - Format Editor](#) for details on the Real Format Editor.

8. To change the format of the variables value, click the **Format** button. The Real Format Editor appears. Specify the number of significant digits, a fixed number of decimal places or have the variable display in exponential form.

9. Make any necessary changes and click **OK**, or click the **Use Default** button for application defaults.



Deleting Variables

1. From the list of available variables, click the variable you want to delete. Select more than one variable at a time by holding down the **CTRL** key, and clicking each variable.
2. Click the **Delete** button.

Deleting a Tab from the Workbook

1. From the list of available Workbook tabs, select the Workbook tab you want to delete.
2. Click the **Delete** button in the Workbook Tabs group.

There is no confirmation message when you delete a Workbook tab and deleted tabs cannot be recovered.

7.23.6 Sorting Information

Workbook tabs can be sorted independently. You can sort the objects on a tab or you can set the order of the variables on a tab.

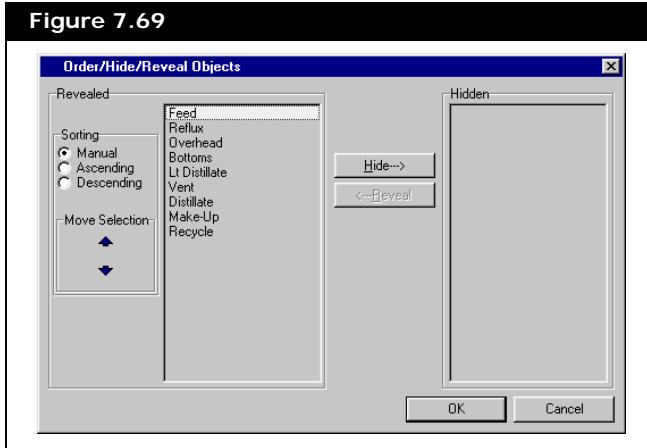
Sorting Objects

To sort the objects of a tab:

1. Open the Order/Hide/Reveal Objects view using one of the following methods:
 - Select the **Order/Hide/Reveal Objects** command From the Workbook Menu.
 - Use the **Order/Hide/Reveal** command in the Object Inspect menu that appears when you right-click any Workbook tabs or cells.

If the Workbook tab is sorted alphabetically, objects continue to be sorted automatically following any ensuing name changes.

- Click the **Order** button in the Object group of the Setup view.



- Use the radio buttons in the Sorting group to specify the sorting method being used.
 - Manual.** From the list of revealed objects, select an object (or multiple objects) using the up and down arrows to manually move the selected object(s) through the list.
 - Alphabetical Ascending.** Sorts the names of the objects in alphabetically ascending order. Objects with numerical names are listed first.
 - Alphabetical Descending.** Sorts the names of the objects in alphabetically descending order. Objects with numerical names are listed last.



Up and down arrows

Hiding & Revealing Objects

To hide an object:

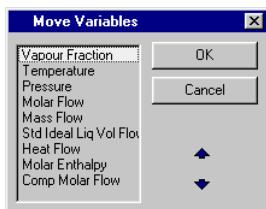
- From the list of revealed objects, select the object(s) you want to hide. Select more than one at a time by holding down the **CTRL** key and clicking each object being selected.
- Click the **Hide** button. The selected object(s) appear(s) in the Hidden list.

To reveal a hidden object:

- From the list of hidden objects, select the object(s) you want to reveal. Select more than one at a time by holding down the **CTRL** key and clicking each object being selected.
- Click the **Reveal** button. The selected object(s) appears in the Revealed list.

Sorting Variables

If variables were added to the Workbook tab as a group (i.e., component molar flows), then you cannot move these individually, but only as a group.



Move Variables view

1. Select **Setup** from the **Workbook** menu. The Setup view appears.
2. From the list of available Workbook tabs, select the Workbook tab with the variables you want to sort.
3. Click the **Order** button in the Variables group. The Move Variables view appears as shown on the left.
4. From the list of available variables, select the variable(s) you want to move. Select more than one at a time by holding down the **CTRL** key and clicking each variable being selected.
5. Click the **Up Arrow** or **Down Arrow** icon.

7.23.7 Exporting/Importing Workbook Tabs

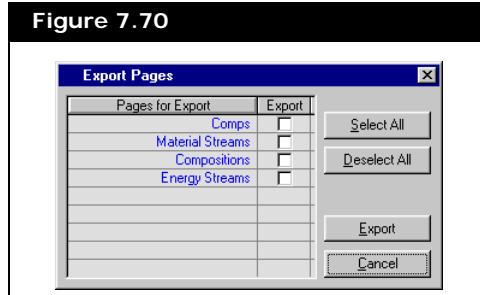
UniSim Design allows you to import and export Workbook information. This same Workbook setup can then be used in other simulation cases.

Exporting a Workbook

1. From the **Workbook-Export** sub-menu, select the **Workbook** command. The Save File view appears.
2. Specify a name and location for your Workbook file.
3. Click **Save**.

Exporting a Workbook Tab

1. From the **Workbook-Export** sub-menu, select the **Pages** command. The Export Pages view appears.



Click the **Select All** button to checked all of the checkboxes.

Click the **Deselect All** button to unchecked all of the checkboxes.

2. Click the Export checkbox by the name of the tab you want to export.
3. Click the **Export** button. The Save File view appears.
4. Specify a name and location for your Workbook file and click **Save**.

Importing a Workbook

1. From the **Workbook-Import** sub-menu, select the **Workbook** command. The Open File view appears.
2. Browse to the location of your Workbook file (*.wrk).
3. Select the file you want to import and click **Open**.

The imported Workbook replaces the existing one.

Importing a Workbook Tab

1. From the **Workbook-Import** sub-menu, select the **Pages** command. The Open File view appears.
2. Browse to the location of your Workbook file (*.wrk).
3. Select the file you want to import and click **Open**.

The imported Workbook tabs are added to the existing Workbook.

7.24 PFD



Any PFD in the simulation can be accessed from any location in the Simulation environment by clicking the PFD icon or using the **CTRL P** hot key.

The Process Flow Diagram (PFD) is the default view when you first enter the Simulation environment. The PFD provides the best representation of the flowsheet as a whole. Using the PFD gives you immediate reference to the progress of the simulation currently being built, such as what streams and operations are installed, flowsheet connectivity, and the status of objects.

The PFD is the default view that appears when you enter the Simulation environment. You can change the Simulation environment default view in your Session Preferences. Refer to [Section 12.2.3 - Desktop Page](#) for additional information.

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 is available so you can reposition streams and operations, resize icons, or reroute streams. All of these tools are designed to simplify the development of a clear and concise graphical process representation.

For information regarding manipulating PFDs, see [Section 10.3 - Editing the PFD](#).

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 and information is automatically updated when changes are made to the process.

There are several ways you can track a specific variable throughout the PFD, including replacing stream name labels or designating a colour to represent a variable range.

Every flowsheet (or sub-flowsheet) has its own PFD, so you can access any flowsheet's PFD from any location. You can also use the multi-flowsheeting architecture to provide clear and concise representations of complex simulations. Instant access to the sub-flowsheet PFD is available through Object Inspect menu of the main flowsheet's PFD.

7.24.1 Custom PFD Notebook

You can customize the PFD into a Notebook by adding new tabs. Using multiple tabs allows for flexibility when dealing with large and complex flow diagrams. For example, one PFD can be used as the main tab for the entire process, while subsequent tabs deal with more specific areas.

Notebooks are also useful when you want each PFD to have a distinct colour scheme for an identical object's setup.

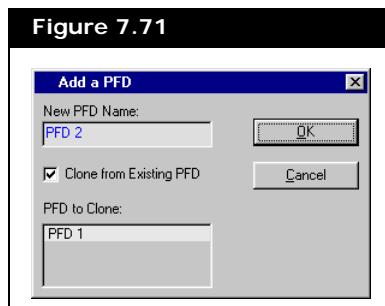
Installing a New PFD

Each PFD is an independent element in that 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 appears on all PFDs.

Cloning a PFD produces an exact duplicate of the selected PFD.

To add a new PFD to the Notebook:

- From the **PFD** menu, select **Add a PFD**. The Add a PFD view appears.



- In the **New PFD Name** field, type the name of your PFD.
- Click the **OK** button.
- If you want your new PFD to be a clone of an existing PFD, check the **Clone from Existing PFD** checkbox. From the list of available PFDs, select the PFD you want to clone and click **OK**. If you just want to create a new blank PFD, ensure the **Clone from Existing PFD** checkbox is unchecked.

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

Deleting a PFD

You cannot recover a deleted PFD.

Unlike the deletion of a single object, deleting a PFD removes it from the Notebook, but does not remove the associated objects from the simulation case.

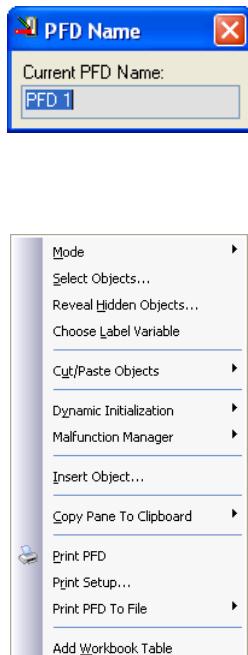
1. Click the PFD tab that you want to delete.
2. Do one of the following:
 - Select **Delete this PFD** from the **PFD** menu.
 - Right-click the PFD tab, and select Delete this PFD.

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

Renaming a PFD

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

1. Make the PFD active by selecting its tab.
2. Select **Rename this PFD** from the **PFD** menu. The PFD Name view appears.
3. In the **Current PFD Name** field, type the new name of the PFD.
4. Click the **Close** icon  to return to the PFD.



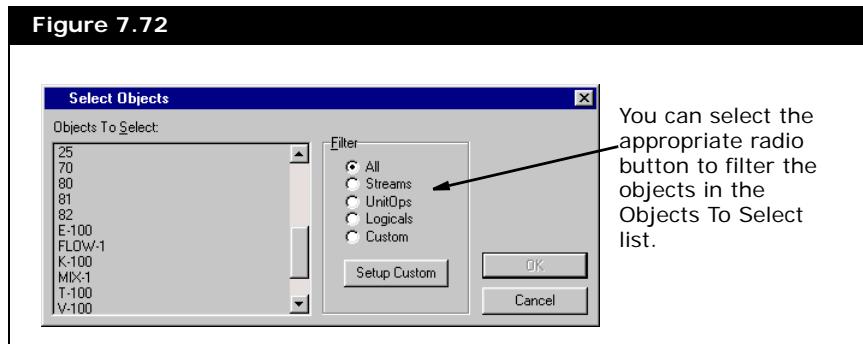
7.24.2 Locating Objects in PFD

The **Select Objects** command in the PFD Object Inspect menu enables you to find objects in the PFD.

To access the locating option in the PFD:

1. Open the PFD of the simulation case.
2. Right-click in any empty area of the PFD to access the Object Inspect menu.
3. Click the **Select Objects** command.

The Select Objects view appears.



4. Select the object you want from the list and click the **OK** button.
The object you are looking for will be selected on the PFD and will have a blinking white frame around it.

7.24.3 Flowsheet Analysis Using the PFD

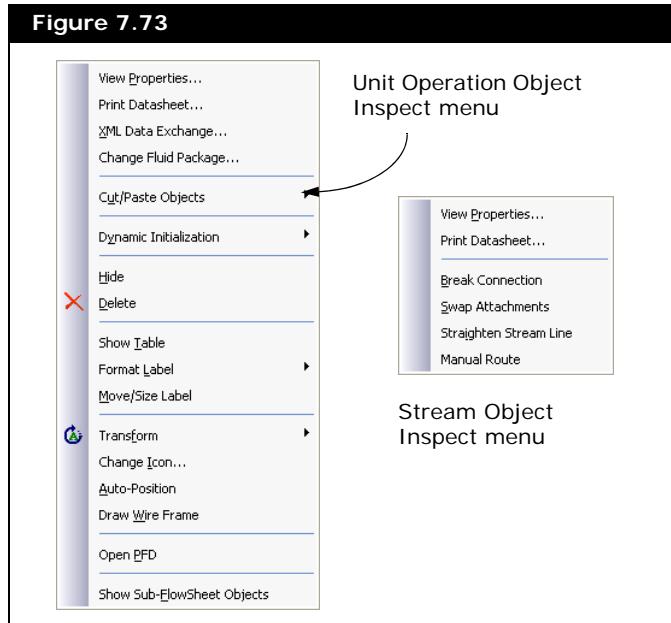
After installing your streams and operations in the PFD, you need to supply specifications. Specifications are entered using the Workbook or an object's property view. Property views are accessed through the PFD, so you can keep the PFD open while supplying the necessary object information. Tables containing specified variables for streams and operations can also be installed in the PFD.

Accessing Stream & Operation Property Views

You can also press the **V** or **E** key to open the property view of a single selected object in the PFD view.

Open the property views directly from the PFD by double-clicking the object icon. Also, you can right-click on an object's icon and select View Properties from the Object Inspect. menu that appears, as shown in the

following figure.

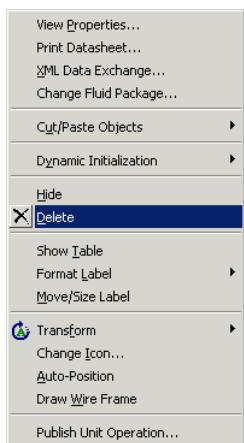


View the property view for a stream by double-clicking the stream icon or by right-clicking any portion of the line that represents the stream and selecting the View Properties command from the Object Inspect menu.

You can also use the Object Status Window to open property views. Move the cursor to the thick border (directly above the status bar at the bottom of the Desktop) and it changes to a vertical line with two arrowheads. Click and hold the mouse button while dragging the cursor upward to expand the Object Status and Trace Windows.

The left pane is the Object Status Window and contains status messages for the various streams and operations. By double-clicking a message, the property view for the associated object appears.

The Object Status Window option is not available after the case solves. At this point, all object status messages are OK and are no longer displayed in the window.



Deleting Streams & Operations

You can delete streams and operations from the case in the PFD by doing either of the following:

- Select the object being deleted and press the **DELETE** key on the keyboard.

- Right-click the object and select **Delete** from the Object Inspect menu, as shown on the left.

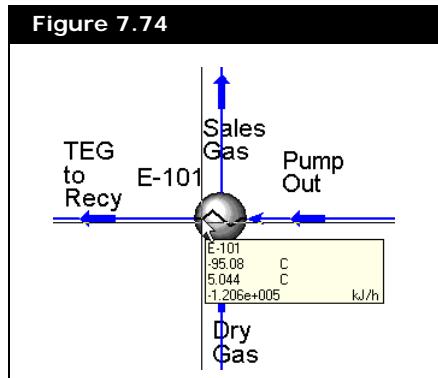
Using either method, you are prompted to confirm the deletion of the object. If multiple objects are being deleted at once, you must confirm each deletion.

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

Fly-by Information

The controls for the Fly-by are found in the Session Preferences. Refer to [Section 12.2.5 - Tool Tips Page](#) for more information.

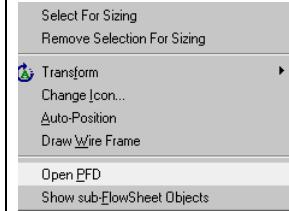
You can view Information related to an object by placing the cursor over its associated icon. A small box listing the object name and the current values of key variables appears. This box is called a Fly-by. The Fly-by of a heat exchanger is shown below.



7.24.4 Access Column or Sub-Flowsheet PFDs

Right-clicking a column (or sub-flowsheet template) in the main PFD displays an Object Inspect menu similar to the one for streams and operations, but with the addition of two options:

- Open PFD command opens the separate column or sub-flowsheet PFD into the case (main) environment.
- Show sub-FlowSheet Objects command displays all sub-flowsheet objects (streams, unit operations, tables, text, etc.) in the main flowsheet. This allows all sub-flowsheet streams and operations to be viewed and accessed from the main PFD without entering the sub-flowsheet environment.

Figure 7.75

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 can become crowded and may need to be re-arranged manually.

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



To hide sub-flowsheet objects displayed in the main PFD, right-click the tip of any sub-flowsheet stream connected to the main flowsheet. The tip of the stream has a small square visible. When you right-click, the Hide sub-FlowSheet Objects command appears. Clicking this command hides all associated sub-flowsheet objects.

Column Sub-Flowsheet

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 has a unique PFD that displays the complete representation of the column flowsheet internal view, including reboilers and condensers.

The Column PFD displays the Columns internal streams, such as Boilup and Reflux. Also, in the Column PFD, the Column stages appear.

When the column PFD is accessed from the main flowsheet, you are not able to modify internal sub-flowsheet connections. You must enter the column sub-flowsheet to perform such tasks as adding or deleting objects and breaking stream connections.

7.24.5 Opening Controller Face Plates



Controllers can also be right-clicked like streams, operations, and columns. The commands available are the same as those associated with the other operations, except for the addition of the Face Plate command. Selecting this command opens the face plate of the selected controller.



Colour Scheme icon

7.24.6 PFD Colour Schemes

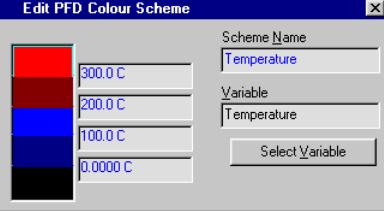
By viewing the colour schemes on the PFD, you can retrieve specific information about your case. The type of information that is available depends on the selected colour scheme.

For example, in default mode, a unit operation can be red, indicating a serious status message associated with the object. The red status can 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 is helpful.

Each PFD can have its own distinct colour scheme.

There are three colour scheme options supplied with UniSim Design:

Scheme	Description
Default Colour Scheme	The colour of unit operations and streams is changed to reflect the status of the object. Unit Ops are red if a serious message is in the Object Status Window, outlined in yellow if a warning message exists, and completely grey if the object has solved. A Stream icon appears light blue if unsolved and dark blue if solved. PFD default colours can be changed on the Colours page of the Session Preferences view. Refer to Section 12.6.1 - Colours Page .

Scheme	Description
HYSIM Colour Scheme	<p>Streams and unit operation icons are shown as wire frames and the colours can be changed. Right-click an object and click the Change Colour command. The colour palette appears and a new colour can be selected. Select an existing colour or click the Define Custom Colours button to customize a colour. After a colour is selected, click the OK button. The new colour for the wire frame appears.</p> <p>Simultaneously change the colour of multiple wire frames by selecting all of the required objects.</p>
Query Colour Scheme	<p>The value of a specified variable can be monitored for all material streams. You can select five colours and an associated variable range for each.</p>  <p>For the example given in the figure above, the top colour (Colour 1) appears for material streams that have a temperature greater or equal to 300°C. Colour 2 represents streams ranging from 200 to 300°C, etc. The last colour (Colour 5) is shown for streams that have temperatures below 0°C. Refer to the following sections for information about working with query colour schemes.</p> <p>The Temperature colour scheme (shown in the Colour Scheme drop-down list when the PFD is accessed), is a Query scheme.</p>

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:



Colour Scheme icon

Figure 7.76



- Click the **Colour Scheme** icon in the toolbar. The PFD Colour Schemes view appears. From the Current Scheme drop-down list, select a colour scheme. Click the **Close** icon to return to the PFD.

When the simulation case is saved, the active colour scheme for each

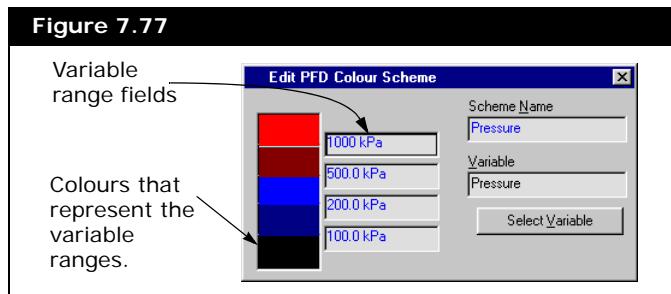
PFD is also stored.

The Delete this Scheme and Edit this Scheme buttons appear for query colour schemes only.

Adding a Query Colour Scheme

You can add a colour scheme that tracks a key material stream variable throughout the PFD.

1. Click the **Colour Scheme** icon in the toolbar. The PFD Colour Schemes view appears.
2. Click the **Add a Scheme** button. The Select Query Variable view appears.
3. From the list of available variables, select the variable you want to monitor. Some variables require a qualifier such as the Comp Mole Frac variable, which requires a component from the list of variable specifics.
4. Click the **OK** button. The Edit PFD Colour Scheme view appears.
5. Input the appropriate values in the variable range fields.



For details about available colour scheme changes, refer to the [Editing a Query Colour Scheme](#) section.

6. To change the colour of the variable range, double-click a colour to access the colour palette. Changes can also be made to the scheme name and to the variable.
7. Click the **Close** icon to return to the PFD Colour Schemes view. The new colour scheme is active.
8. Click the **Close** icon to return to the PFD.

Deleting a Query Colour Scheme

Edit and Delete buttons appear only when a Query Colour Scheme is selected.

1. Click the **Colour Scheme** icon in the toolbar. The PFD Colour Schemes view appears.
2. Select a colour scheme from the Current Scheme drop-down list.

3. Click the **Delete this Scheme** button.

Ensure the correct scheme is selected before deleting. You cannot recover deleted schemes.

Editing a Query Colour Scheme

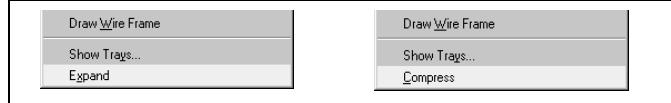
1. Click the **Colour Scheme** icon in the toolbar. The PFD Colour Schemes view appears.
2. Select a colour scheme from the Current Scheme drop-down list.
3. Click the **Edit this Scheme** button. The Edit PFD Colour Scheme view appears as shown previously in [Figure 7.77](#).
4. In this view, edit the scheme name, query variable, variable ranges, and variable range colours. Refer to [Adding a Query Colour Scheme](#) in the previous section for more information.
5. After all changes are made to the query colour scheme, click the **Close** icon  to return to the PFD Colour Schemes view.
6. Click the **Close** icon  on the PFD Colour Schemes view to return to the PFD.

Only one colour can be changed at a time.

7.24.7 Column Tray Section Display

In the column sub-flowsheet, you can right-click the individual components of the Column, (i.e., Tray Section, Condenser, and Reboiler). The Object Inspect menu of the Tray Section provides two additional commands: Show Trays and Compress/Expand as shown below.

Figure 7.78



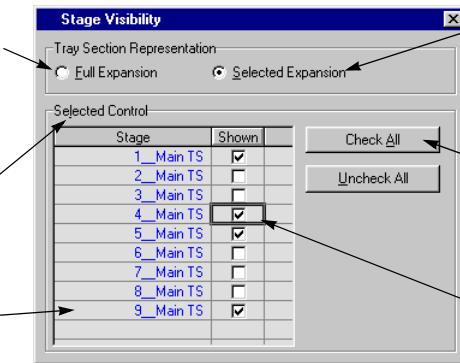
Click the Show Trays command to open the Stage Visibility view.

Figure 7.79

When this radio button is selected, the Column appears showing each tray. For instance, a 10-tray column shows trays 1 through 9 on the PFD.

This group can only be accessed when the Selected Expansion radio button is selected.

UniSim Design always draws the column showing the first and last stages, as well as feed and draw stages.



When this radio button is selected, you can compress the column tray section displayed on the PFD.

Instead of individually checking all the checkboxes, use these buttons to check or uncheck all the stages in the Column.

Check the checkbox for each tray you want to display on the PFD.

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

Click the Expand option to expand the column to the full size, showing all trays. Click the Compress option to compress the column to the settings in the Stage Visibility view. Only the selected trays will appear.

7.24.8 PFD Tables

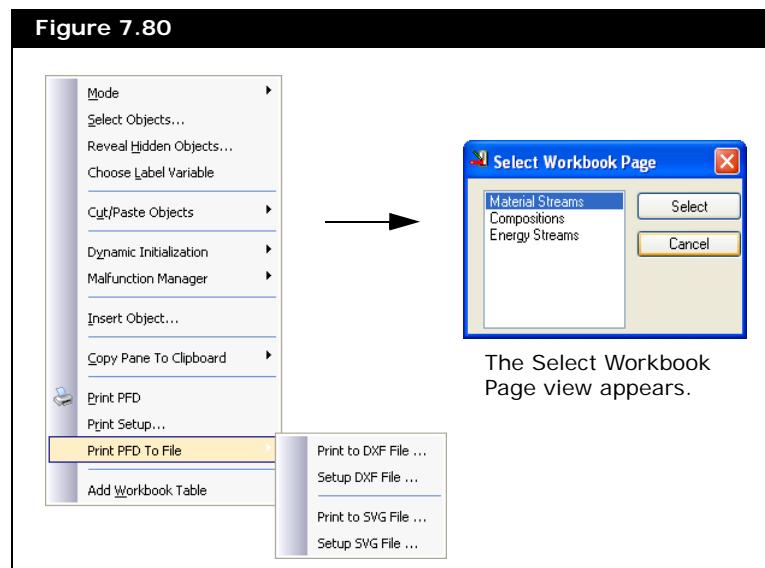
Workbook Table

You can display any full Workbook tab as a table on the PFD except for the Unit Ops tab. All information on the Workbook tab is shown in the table and is automatically updated when changes occur in the flowsheet.

Add a Workbook Table to the PFD

1. Right-click to open the PFD Object Inspect menu and select **Add Workbook Table**. The Select Workbook Page view appears.

2. Click the name of the tab being added, then click the **Select** button as shown below. After the Workbook table is added, click and drag it to the necessary location.



When you right-click a Workbook table, the Object Inspect menu shows the following options:

- **Hide.** Hides the Workbook table.
- **Change font.** Changes the font of the text in the table.
- **Change colour.** Opens the colour palette so you can change the text and table border colours.

Object Variable Table

Install a table for any object in the PFD by right-clicking the stream or operation and clicking the Show Table command as shown on the left. Each object type has a default variable list associated with it. The table shown below is for a Separator.

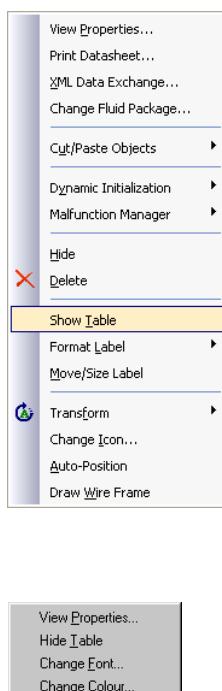


Table Object Inspect menu

You can modify the list of displayed variables as well as the table appearance by right-clicking to access the table's Object Inspect menu.

Figure 7.81

The screenshot shows the 'Flash Tk' table for a Separator object. The table has four columns: 'Variable', 'Value', 'Type', and 'Unit'. The data is as follows:

Variable	Value	Type	Unit
Vessel Temperature	134.7	F	
Vessel Pressure	90.00	psia	
Liquid Molar Flow	4295.	lbmole/hr	
Duty	0.0000	Btu/hr	

The menu contains the following options.

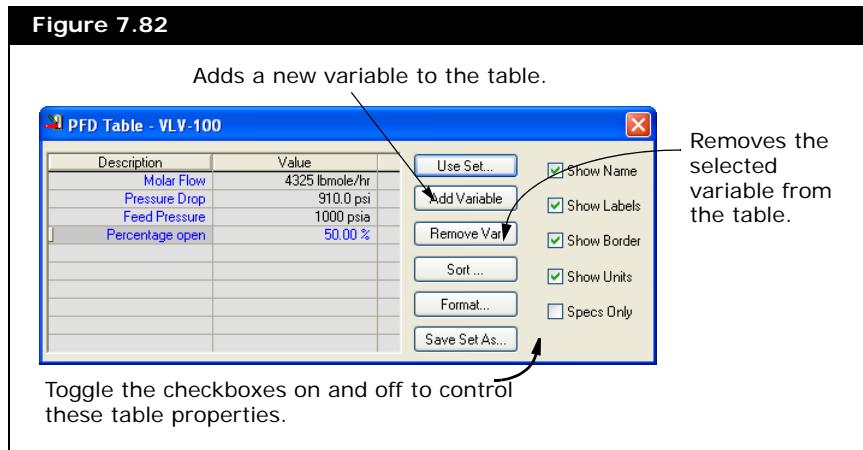
Table Menu Item	Description
View Properties	Access the various properties available for the tables. Refer to the Table Properties section for more information.
Hide Table	Temporarily hides the table on the PFD. When the table is revealed, it is shown as it was before it was hidden.
Change Font	Change the font for the text in the table.
Change Colour	Open the colour palette, and change the colour of the Table text, and the Table outline.

Table Properties

Open the PFD Table view by double-clicking on the table.

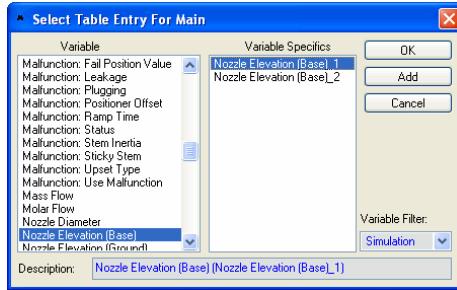
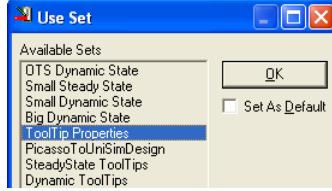
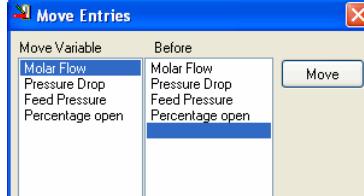
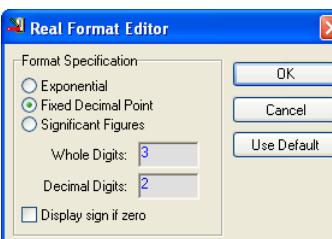
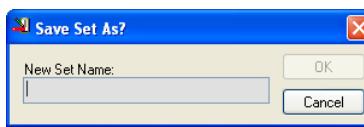
If you are having trouble selecting a table, ensure you are not in Attach mode.

A default variable set is provided for each stream and operation type. To modify this list, right-click the table and select the View Properties command.



Use the buttons available in the view to modify the variable set for the

table. Refer to the following table for the description of each button:

Button	Button Usage	Views/Remarks
Add Variable	<p>To add a variable to the table:</p> <ol style="list-style-type: none"> 1. Click the Add Variable button. The Variable Navigator view appears. 2. From the list of available variables, select the variable you want to add to the table. 3. Make any required changes to the variable description in the Description field. 4. Click the OK button to return to the PFD Table view. <p>The Add Variable option lets you add only one variable at a time to the table.</p>	
Remove Var	<p>Removes variables from the table. Select the variable(s) to remove from the table, and click the Remove Var button.</p> <p>(Select more than one variable at a time by holding down the CTRL key, and then clicking each variable you want to select.)</p>	<p>You are not prompted to confirm the deletion of variables from the table.</p>
Use Set	<p>Default variable sets are provided and accessed by clicking the Use Set button. The list of variable Sets differs, depending on the object type (stream, unit operation, column, or controller). To change to a default variable set, select variable set from the list and click the OK button.</p>	 <p>The sets shown above applies to a Separator.</p>
Sort	<p>Use to reorganize variables in the table. Select the item(s) being moved in the Move Variable field. In the Before field, select the variables that are in front of the variables being moved and click the Move button.</p> <p>(Select more than one variable at a time by holding down the CTRL key, and then clicking each variable you want to select.)</p>	
Format	<p>Change the numeric format of a value. Select the variables being changed from the table. Click the Format button. The Real Format Editor appears. In the Format Specification group, select one of the following radio buttons: Exponential, Fixed Decimal Point, or Significant Figures. Each selection requires you to specify information to fully define the selection. Click the OK button.</p>	
Save Set As	<p>Specify the New Set Name and click the OK button.</p>	

The Specs Only button is turned off by default. Click this checkbox to

show all entries that are currently not specified.

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 example, 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.

7.24.9 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 lets you separate the PFD view into a maximum of four panes. Each pane contains all the information concerning the PFD (operations, connections, etc.), but operates independently in 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. The PFD appears with two borders. The manipulation areas are located along the inner border. You can split the PFD once vertically and once horizontally, for a maximum of four panes. Each new view created has its own zoom and scroll buttons. This enables the various views to be at different zoom levels or locations in the overall PFD.



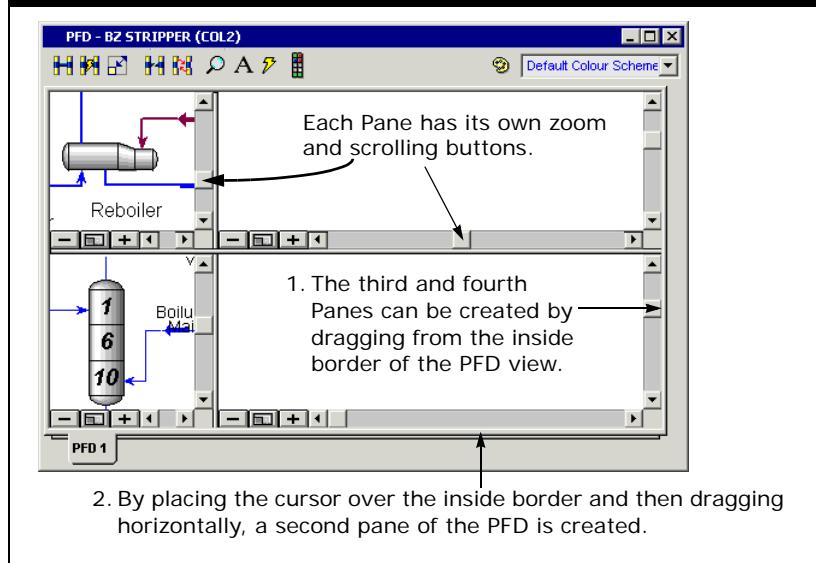
Multi-Pane Sizing Tool

When the cursor is over the inside border of the PFD, the pointer changes into the multi-pane sizing tool. The symbol is rotated 90 degrees for a horizontal split.

The Status Bar in the bottom of the screen indicates the way the split

occurs.

Figure 7.83



There is no requirement for how 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 on different sections of the PFD in each pane.

The splitting of PFD panes becomes useful when the PFD is complex and you cannot view it entirely without making it very small. When working in split panes, remember that all the panes interact with each other. This enables you to connect an operation or stream in one pane to an operation or stream in another. Changes made in one pane are made in the overall PFD.

Resizing or Closing Split Panes



Multi-Pane Sizing Tool cursor

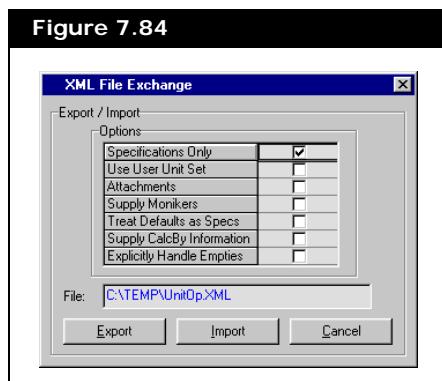
After panes are created, they can be resized or closed. To resize or close a vertically split pane, place the cursor on the right side of the vertical scroll bar (in the split pane). The Multi-Pane Sizing Tool cursor appears. Drag it left or right to change the existing view. Dragging to the extreme right closes the right split pane.

The procedure is the same for a horizontally split pane, except 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 view closes the

bottom split pane.

7.24.10 Exchanging XML Files

When you right-click any object on the PFD, the Object Inspect menu appears. When you select the XML Data Exchange command, the XML File Exchange view appears.



Select the information you want to export or import to the XML file by clicking the corresponding checkboxes. You can change the file name and location by entering the new name and file path in the File field.

Click the Export button to export the file, the Import button to import the file, or the Cancel button to close the view without completing the action.

7.25 Column

This menu item only appears inside the Column environment. The options under Column are as follows:

Column	
Column Runner...	Ctrl+T
Run	F6
Reset	Shift+F6

Refer to [Chapter 8 - Column](#) in the [UniSim Design Operations Guide](#) for more information.

Command	Description
Column Runner	View the Column Runner.
Run	Start the Column Solver.
Reset	Reset the Column Solver.

Refer to [Chapter 14 - Utilities](#) in the [UniSim Design Operations Guide](#) for detailed information about the individual utilities.

7.26 Utilities

The utilities available in UniSim Design are a set of useful tools that interact with your process, providing additional information or analysis of streams or operations. A utility becomes a permanent part of the

flowsheet, automatically recalculating when conditions change in the stream or operation that it is attached to. You can access utilities using any of the following methods:

- Select the **Utilities** command from the **Tools** menu.
- Press the **CTRL U** hot key.
- Click the **Create** button on the **Utilities** page of the **Attachments** tab of a Stream property view.

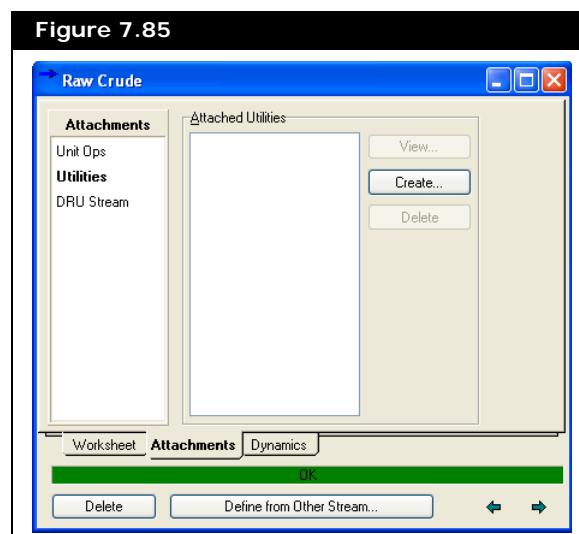
7.26.1 Adding Utilities

There are two ways to add utilities: add the utility in the Stream property view or in the Available Utilities view.

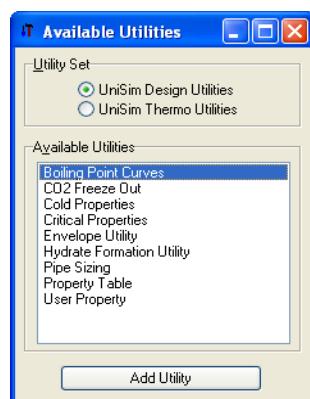
Using the Stream Property View

Not all utilities can be accessed through the Stream property view.

1. Open the property view for the stream, and go to the **Utilities** page of the **Attachments** tab.



2. Click the **Create** button. The Available Utilities view appears as shown on the left.
3. From the list of available utilities, select the utility you want to add.
4. Click the **Add Utility** button. The property view for the selected utility appears.
5. Define the utility as required.



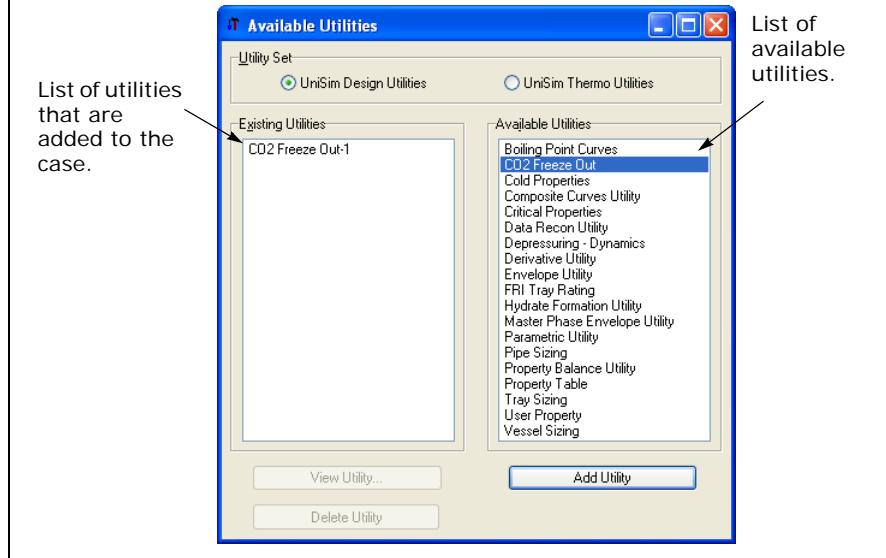
Available Utilities view for stream

Using the Flowsheet

1. Select the **Utilities** command from the **Tools** menu. The Available

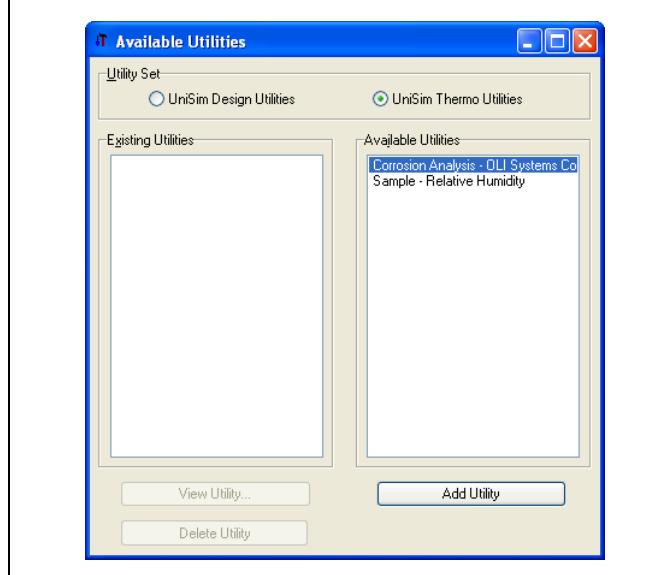
Utilities view appears.

Figure 7.86



2. There are two categories of utilities, one is UniSim Design utilities, the other is UniSim Thermo Utilities. User can chose from the categories by click the radio button.
 - UniSim Design Utilities - UniSim Design native utilities
 - UniSim Thermo Utilities - Plug in utilities (as COM object), mostly from third party

Figure 7.87



3. From the list of available utilities, select the utility you want to add.

4. Click the **Add Utility** button. The property view for the selected utility appears.
5. Define the utility as required.

7.26.2 Viewing Utilities

The property view for a Utility can remain open independently of the stream to which it is attached.

There are two ways to view utilities: from the Stream property view or the Available Utilities view.

Using the Stream Property View

1. Open the property view for the stream.
2. Click the **Attachments** tab and select the **Utilities** page.
3. From the list of attached utilities, select the utility you want to edit.
4. Click the **Edit** button. The property view for the selected utility appears.

Using the Flowsheet

1. Select the **Utilities** command from the **Tools** menu. The Available Utilities view appears.
2. From the list of attached utilities, select the utility you want to edit.
3. Click the **View Utility** button. The property view for the selected utility appears.

7.26.3 Deleting Utilities

There are two ways to delete the utilities: delete the utility in the Stream property view or in the Available Utilities view.

Using the Stream Property View

1. Open the property view for the stream.
2. Click the **Attachments** tab, then select the **Utilities** page.
3. From the list of attached utilities, select the utility you want to delete.
4. Click the **Delete** button. You are not prompted to confirm the deletion of a utility.

Using the Flowsheet

1. Select the **Utilities** command from the **Tools** menu. The Available

Utilities view appears.

2. From the list of attached utilities, select the utility you want to delete.
3. Click the **Delete Utility** button. You are not prompted to confirm the deletion of a utility.

7.27 Simulation Balance Tool

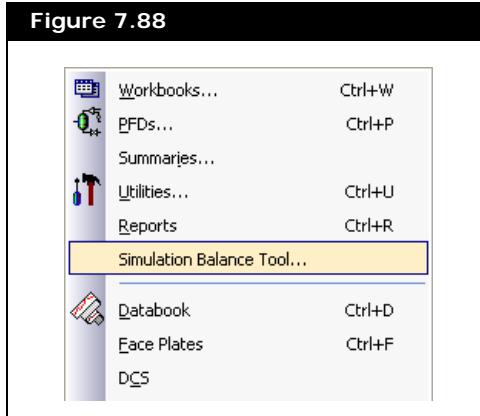
The Simulation Balance Tool provides a mechanism to check for material and energy balances within the UniSim Design flowsheets as a means to gain a rapid overview of all entities which could be sources of flowsheeting errors although a flowsheet has status converged.

You can select to validate any combination of mass, mole, heat and component balances and using absolute and relative tolerances.

7.27.1 Accessing the Simulation Balance Tool

There are two ways to access the Tool:

- Via the Tools menu - from the Tools menu, select Simulation Balance Tool

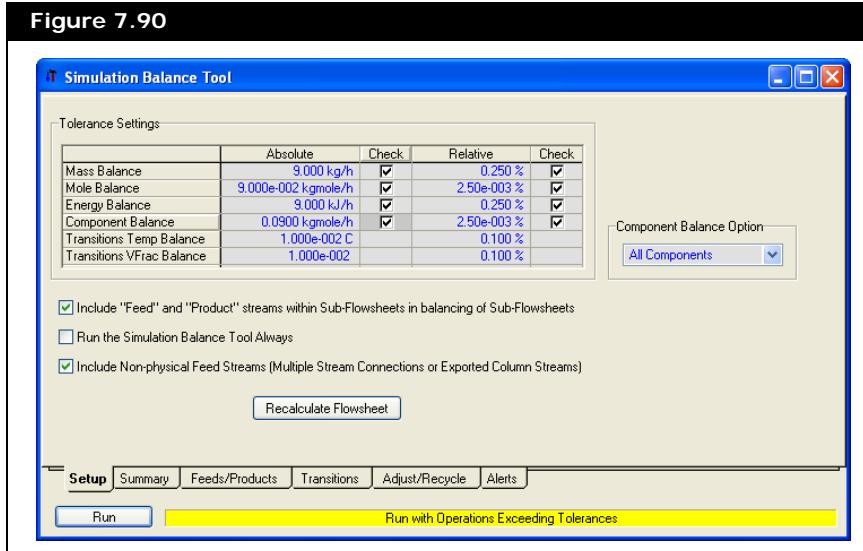


- Via the status bar - locate the Overall Mass Balance Monitor status bar on the bottom right of UniSim Design and double-click on it



7.27.2 Setup Tab

On the Setup page, you can set the parameters for the Simulation Balance Tool as well as perform other functions.



Setting the Simulation Balance Tool Parameters

1. In the **Tolerance Settings** matrix, specify the tolerances and balances to use in the validation.
2. **Component Balance Option** - shows up when Component Balance is checked. Select either all components or a single component to perform the component balancing.
3. Check to **Include “Feed” and “Product” streams within Sub-Flowsheets in balancing of Sub-Flowsheets**, if desired.
4. Check to **Run the Simulation Balance Tool Always**, if desired. When checked, the Tool automatically runs on changes to the flowsheets. If not checked, the Run button must be pressed to re-calculate.
5. Check to **Include Multiple Stream Connections** such as non-physical feed streams (multiple stream connections or exported column streams), if desired.
6. You can also perform the following functions:
 - Press the Recalculate Flowsheet button to force the flowsheet to recalculate.
 - Press the Run button to run the Simulation Balance Tool.

7.27.3 Summary Tab

The Summary tab consists of two pages selectable via a radio button:

- General Summary page
- Detailed Summary page

General Summary

The General Summary page lists all operations that exceed the specified tolerances.

Figure 7.91

The table gives the following information about the operations:

- **Flowsheet** - the operation's parent flowsheet
- **Type** - the UniSim Design object type of the operation
- **Name** - the operation name. You can double-click on the name to bring up the operation view.
- **Mass Error** - the mass balance error in both absolute and percentage values
- **Mole Error** - the molar balance error in both absolute and percentage values
- **Energy Error** - the heat balance error in both absolute and percentage values
- **Comp Error** - the component molar balance error in both absolute and percentage values
- **Comp Name** - the name of the component with the highest molar balance error

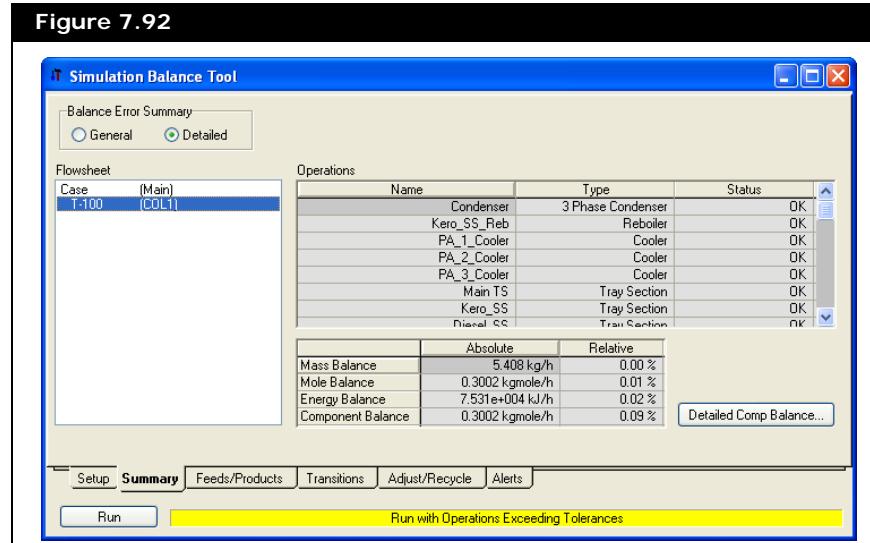
There are also columns with arrows (' , <->) indicating balance errors that exceed the specified tolerances.

Grouping/Sorting

The table of results can be grouped by Flowsheet and Type as well as sorted based on the other columns. The grouping and sorting by name is done in ascending order while the rest of the sorting is in descending order.

Detailed Summary

The Detailed Summary page gives a unit by unit balance summary in both tolerance bases.



To view a particular unit operation summary, select the Flowsheet and Operations and the balance summary table will update appropriately. Double-clicking on the Name of the Operation will bring up the operation property view.

Detailed Comp Balance - opens up the Detailed Component Balance Summary view displaying either a single component or all the components (depending on the Component Balance Option selection)

along with their molar balance errors in absolute and relative bases.

Figure 7.93

Component Name	Absolute Error [kgmole/h]	Relative Error [%]
Methane	0.0000	0.00
Ethane	0.0000	0.00
Propane	0.0000	0.00
i-Butane	0.0000	0.00
n-Butane	0.0000	0.00
H ₂ O	0.3002	0.09
NBP[0]49*	0.0000	0.00
NBP[0]79*	0.0000	0.00
NBP[0]111*	0.0000	0.00
NBP[0]144*	0.0000	0.00
NBP[0]76*	0.0000	0.00
NBP[0]208*	0.0000	0.00
NBP[0]240*	0.0000	0.00
NBP[0]272*	0.0000	0.00
NBP[0]304*	0.0000	0.00

7.27.4 Feeds/Products Tab

The Feeds/Products tab contains the overall mass, mole and heat balance errors in both absolute and relative units, a list of feed streams and a list of product streams along with their flow information.

Figure 7.94

Feed Streams					
Flowsheet	Name	Type	Mass Flow [kg/h]	Mole Flow [kgmole/h]	Heat Flow [kJ/h]
Main	Raw Crude	Material	5.829e+005	2826	-9.741e+008
Main	Main Steam	Material	3402	188.8	4.454e+007
Main	Diesel Steam	Material	1361	75.54	-1.790e+007
Main	AGO Steam	Material	1134	62.95	-1.492e+007
Main	Crude Duty	Energy	<empty>	<empty>	1.844e+008
Main	O-Trim	Energy	<empty>	<empty>	7.496e+007
Total			5.8875e+005	3153.7	-7.8426e+008

Product Streams					
Flowsheet	Name	Type	Mass Flow [kg/h]	Mole Flow [kgmole/h]	Heat Flow [kJ/h]
Main	Residue	Material	2.810e+005	644.0	-3.813e+008
Main	Oil Gas	Material	0.0000	0.0000	0.0000
Main	Waste Water	Material	5728	318.0	-9.034e+007
Main	Naphtha	Material	1.115e+005	1256	-2.439e+008
Main	Kerosene	Material	5.155e+004	327.5	-8.930e-007
Main	Diesel	Material	1.117e+005	516.8	-1.857e+008
Total			5.8875e+005	3154.0	-7.8434e+008

Overall Balance Errors:

- Mass Error: 5.408 kg/h (Green)
- Mole Error: 0.3002 kgmole/h (Grey)
- Energy Error: 7.830e+004 kJ/h (Grey)
- Component Error: 0.3002 kgmole/h (Grey)
- Comp Balance Summary...

Setup Summary **Feeds/Products** Transitions Adjust/Recycle Alerts

Run Needs Recurring

The Overall Balance Errors are displayed in different colours:

- **Green** - the errors are within the specified tolerances
- **Red** - the errors exceed the specified tolerances

The Feeds and Products tables list the following information about the streams:

- **Flowsheet** - the stream's parent flowsheet

- **Name** - the name of the stream. Double-clicking the name will bring up the stream's property view.
- **Type** - the UniSim Design object type of the stream
- **Mass Flow** - the stream's mass flow rate
- **Mole Flow** - the stream's molar flow rate
- **Heat Flow** - the stream's heat flow rate

There is also a row of total flows for the Feed streams and a row of total flows for the Product streams.

Comp Balance Summary - opens up the Component Balance Summary view displaying either a single component or all the components (depending on the Component Balance Option selection) along with their molar balance errors in absolute and relative bases. You can also sort based on the component name, absolute and relative error.

Figure 7.95

Component Name	Absolute Error [kgmole/h]	Relative Error [%]
H2O	0.3002	0.09
Methane	0.0000	0.00
Ethane	0.0000	0.00
NBP[0]368*	0.0000	0.00
NBP[0]304*	0.0000	0.00
NBP[0]111*	0.0000	0.00
NBP[0]336*	0.0000	0.00
NBP[0]528*	0.0000	0.00
NBP[0]560*	0.0000	0.00
NBP[0]830*	0.0000	0.00
NBP[0]1124*	0.0000	0.00
NBP[0]464*	0.0000	0.00
NBP[0]433*	0.0000	0.00
NBP[0]1062*	0.0000	0.00
NBP[0]624*	0.0000	0.00

7.27.5 Transitions Tab

The Transitions page summarizes all interconnecting streams and transition points that have exceeded the specified tolerances. The interconnecting streams and transition points include external to internal streams at flowsheet boundaries, stream cutters and recycles.

Click the Hide Ignored checkbox to hide everything with status-ignored.

Figure 7.96

Simulation Balance Tool

Flowsheet transitions exceeding specified tolerances Hide Ignored Normal Transpose

Parent Sheet	Ext Stream	SubFlwshl	Int Stream	Unit Op	Type	Trans Basis	Status	Mass Error [kg/h]	Mass Error [%]	Mole Error [kgmole/h]
Main	Atm Feed	COL1	Atm Feed	T-100	Column S	T-P Fla	OK	0.0000	0.0000	0.000
Main	Naphtha	COL1	Naphtha	T-100	Column S	T-P Fla	OK	0.0000	0.0000	0.000

Setup Summary Feeds/Products **Transitions** Adjust/Recycle Alerts Run Run with Operations Exceeding Tolerances

Figure 7.97

Simulation Balance Tool

Flowsheet transitions exceeding specified tolerances Hide Ignored Normal Transpose

Parent Sheet	Main	Main
External Stream	Atm Feed	Naphtha
SubFlowsheet	COL1	COL1
Internal Stream	Atm Feed	Naphtha
Unit Op	T-100	T-100
Type	Column Sub-Flo	Column Sub-Flo
Transfer Basis	T-P Flash	T-P Flash
Status	OK	OK
Mass Error [kg/h]	0.0000	0.0000
Mass Error [%]	0.0000	0.0000
Mole Error [kgmole/h]	0.0000	0.0000
Mole Error [%]	0.0000	0.0000
Heat Error [kJ/h]	32.79	46.52
Heat Error [%]	4.152e-006	1.907e-005
Temp Error [C]		
Temp Error [%]		
VFrac Error	2.772e-007	2.013e-006
VFrac Error [%]	4.580e-005	100.0
Comp Error [kgmole/h]		

Setup Summary Feeds/Products **Transitions** Adjust/Recycle Alerts Run Run with Operations Exceeding Tolerances

The table contains the following columns:

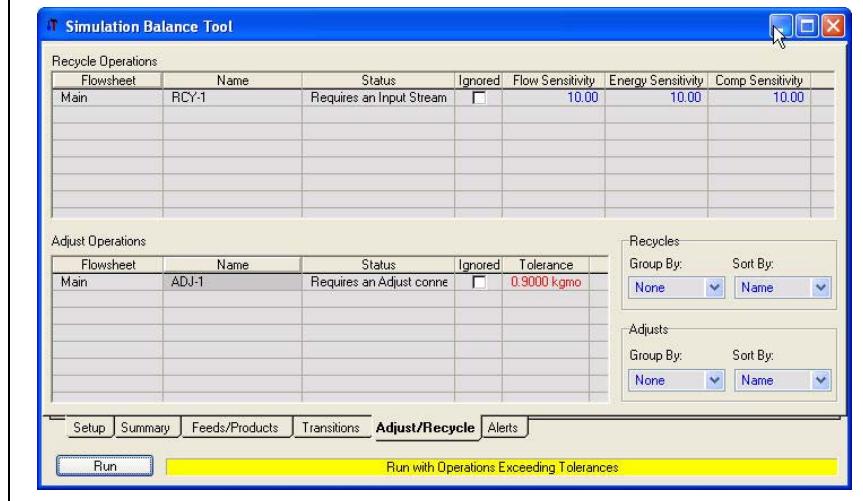
- **Parent Sheet** - the parent flowsheet of the external stream
- **External Stream** - the name of the external stream. Double-clicking will bring up the stream's property view.
- **SubFlowsheet** - the parent flowsheet of the internal stream
- **Internal Stream** - the name of the internal stream. Double-clicking will bring up the stream's property view.
- **Unit Op** - the name of the unit operation that the interconnecting stream is attached to. Double-clicking will bring up the operation's property view.
- **Type** - the UniSim Design object type of the unit operation

- **Transfer Basis** - the transfer basis used in the interconnecting stream
- **Status** - the solving status of the operation
- **Mass Error** - the mass balance error in both absolute and percentage values
- **Mole Error** - the molar balance error in both absolute and percentage values
- **Heat Error** - the heat balance error in both absolute and percentage values
- **Temp Error** - the temperature balance error in both absolute and percentage values
- **VFrac Error** - the vapour fraction balance error in both absolute and percentage values
- **Comp Error** - the component molar balance error in both absolute and percentage values
- **Comp Name** - the name of the component with the highest molar balance error

7.27.6 Adjust/Recycle Tab

The Adjust/Recycle page lists all recycle and adjust operations along with their statuses and defined tolerances.

Figure 7.98



The following information is displayed:

- **Flowsheet** - the parent flowsheet of the operation
- **Name** - the name of the operation. Double-clicking brings up the operation's property view.
- **Status** - the solving status of the operation
- **Ignored** - toggle to ignore the calculation of the unit operation
- **Flow Sensitivity** - the specified flow sensitivity value of the recycle operation

- **Energy Sensitivity** - the specified energy sensitivity value of the recycle operation
 - **Comp Sensitivity** - the specified component sensitivity value of the recycle operation
 - **Tolerance** - the defined tolerance of the adjust operation

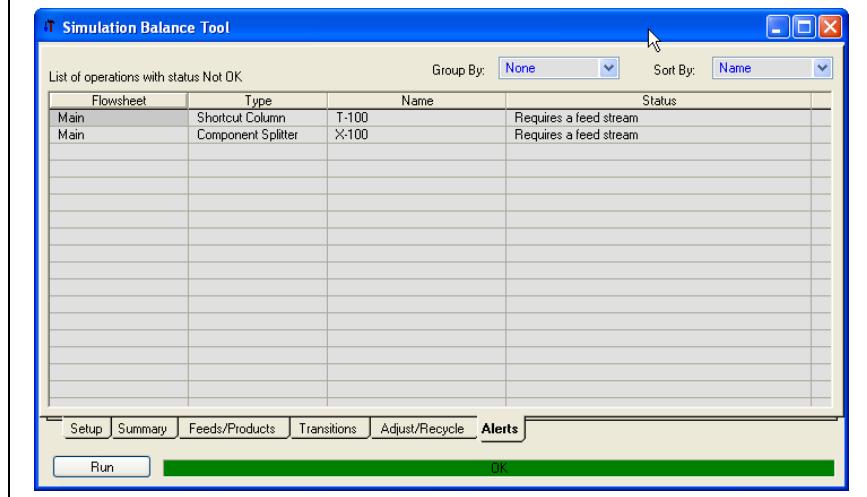
Grouping/Sorting

The recycles and adjusts tables can be grouped by Flowsheet as well as sorted by Name and Status. The grouping and sorting is done in ascending order.

7.27.7 Alerts Tab

All operations with a solve status that is Not OK are populated on this page.

Figure 7.99



The table contains the following columns:

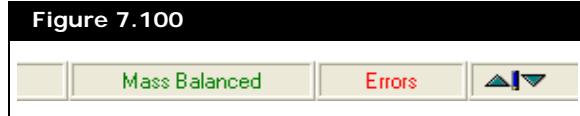
- **Flowsheet** - the parent flowsheet of the operation
 - **Name** - the name of the operation. Double-clicking brings up the operation's property view.
 - **Type** - the UniSim Design object type of the operation
 - **Status** - the solving status of the operation

Grouping/Sorting

The table of results can be grouped by Flowsheet and Type as well as sorted based on Name and Status. The grouping and sorting is done in ascending order.

7.27.8 Status Bar Monitoring

The enhancement to the status bar includes helpful information on the status of the overall mass balance as run by the Simulation Balance Tool and an overall status monitor that summarizes the status window list.



Overall Mass Balance Monitor

The overall mass balance monitor gives the status of the overall mass balance when the Simulation Balance Tool is run. Double-clicking on this will bring up the Simulation Balance Tool property view.

Overall Status Monitor

This status monitor gives a summary of the flowsheeting status as captured in the Status Window List.

8 UniSim Design Objects

8.1 Installing Objects	3
8.1.1 Install Objects Using the Object Palette.....	3
8.1.2 Install Objects Using the Flowsheet Menu.....	5
8.2 Defining Objects	5
8.2.1 Defining a Material Stream.....	5
8.2.2 Defining an Energy Stream.....	6
8.2.3 Defining a Separator	6
8.2.4 Defining a 3-Phase Separator	7
8.2.5 Defining a Tank.....	7
8.2.6 Defining a Single Outlet Vessel.....	8
8.2.7 Defining a Cooler/Heater.....	8
8.2.8 Defining an LNG Exchanger	9
8.2.9 Defining a Heat Exchanger	10
8.2.10 Defining an Air Cooler	10
8.2.11 Defining a Pump	11
8.2.12 Defining a Compressor/Expander.....	11
8.2.13 Defining a Compressible Gas Pipe.....	12
8.2.14 Defining a Pipe Segment	13
8.2.15 Defining a Valve	13
8.2.16 Defining a Relief Valve	14
8.2.17 Defining a Mixer.....	14
8.2.18 Defining a Tee.....	15
8.2.19 Defining a Simple Solid Separator.....	15
8.2.20 Defining a Cyclone.....	16
8.2.21 Defining a Hydrocyclone	16
8.2.22 Defining a Rotary Vacuum Filter	17
8.2.23 Defining a Baghouse Filter	17
8.2.24 Defining a Screen	18
8.2.25 Defining a Conveyor	18
8.2.26 Defining a Crusher	19
8.2.27 Defining a CSTR.....	19
8.2.28 Defining a PFR	20
8.2.29 Defining a Gibbs Reactor	21
8.2.30 Defining an Equilibrium Reactor.....	22
8.2.31 Defining a Conversion Reactor.....	22
8.2.32 Defining a Neutralizer	23
8.2.33 Defining a Crystallizer	24
8.2.34 Defining a Precipitator	24
8.2.35 Defining a Distillation Column	25

8.2.36 Defining a Refluxed Absorber Column.....	26
8.2.37 Defining an Absorber Column.....	27
8.2.38 Defining a Reboiled Absorber Column.....	28
8.2.39 Defining a Liquid-Liquid Extractor	29
8.2.40 Defining a Three Phase Distillation Column	30
8.2.41 Defining a Component Splitter	31
8.2.42 Defining a Short Cut Distillation Column	31
8.2.43 Defining an Adjust.....	32
8.2.44 Defining a Set.....	33
8.2.45 Defining a Recycle	33
8.2.46 Defining a PID Controller.....	33
8.2.47 Defining a Selector Block.....	34
8.2.48 Defining a Balance.....	34
8.2.49 Defining a Digital Point.....	34
8.2.50 Defining a Transfer Function Block	35
8.2.51 Defining an MPC Controller	35
8.2.52 Defining a Centrifuge	35
8.2.53 Defining a Virtual Stream	36

8.1 Installing Objects

Objects are used to build your simulation within the Simulation environment. The objects in UniSim Design are streams, unit operations or logical operations.

To enter the Simulation environment click the Enter Simulation Environment button or Return to Simulation Environment button in the Basis environment.

Refer to [Section 7.23.2 - Installing Streams or Operations](#) for more information about the Workbook.

Use one of the following methods to install an object in your simulation:

- Object Palette
- Workbook
- Flowsheet-Add Stream/Add Operations command

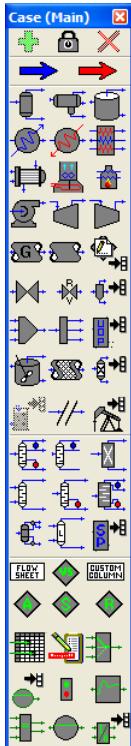
8.1.1 Install Objects Using the Object Palette

To access the Object Palette (shown on the left), select the Object Palette command from the Flowsheet menu or press **F4**.

In the main flowsheet or template sub-flowsheet, all available operations are accessible through the palette except those specifically used with Columns (tray sections, reboilers, etc.). A modified palette appears when you are inside the column sub-flowsheet.

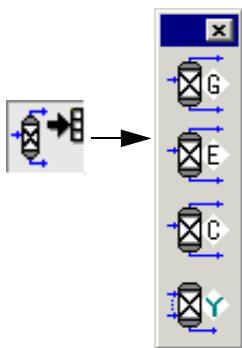
The Object Palette is organized into the following categories from top to bottom:

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



General Buttons

Icons on the object palette with an arrow pointing to the right are



The General Reactors icon opens this sub-palette.

general icons (for example, the Solid Ops and General Reactors icons) that contain sub-palettes. These sub-palettes display icons for more specific unit operations.

As shown on the left, you can install a Gibbs Reactor, an Equilibrium Reactor, or a Conversion Reactor from the sub-palette.

Each operation has an icon and when you place the cursor over it, a fly-by description of the operation appears below the cursor and in the Status Bar.

You can install a single stream, unit operation, or logical operation from the palette by double-clicking the icon for the object you want to install. The object appears in the PFD and the object's property view is opened.



Add icon

You can also use the Add icon at the top of the palette to install objects.

1. Click the icon for the object you want to install.
2. Click the **Add** icon to insert the object in the PFD and open the object's property view.



Lock icon

You can install same objects multiple times by using the Lock feature.

1. Click the **Lock** icon at the top of the palette.
2. Click the icon of the object you want to install.
3. Click the **Add** icon to install the object. With the lock feature active, you can add as many of the selected object as required without having to click the object icon.
4. To switch objects when in locked mode, click the Cancel icon or click a different icon.
5. To stop the Lock function, click the **Lock** icon again.



Cancel icon

You can also use the drag-and-drop method of installing objects.

1. Click the icon of the object you want to install.
2. Move the cursor into the PFD. Your cursor changes to an arrow with a '+' and a white outline of the object.
3. In the PFD, click where you want to install the object. The object appears in the PFD (but the object's property view does not automatically open).

8.1.2 Install Objects Using the Flowsheet Menu

Installing Streams

Press the **F11** hot key to quickly add a stream to the simulation.

Select the Add Stream command from the Flowsheet menu to install a stream in the PFD. When the stream is added to the PFD, the stream property view opens.

Material streams can only be added using the Add Stream command in the Flowsheet menu whereas energy streams can only be added using the Object Palette.

Installing Operations

Press the **F12** hot key to quickly open the UnitOps view.

Use the radio button in the Categories group to filter the list of available unit operations making it easier to find the operation you want to add.

1. Select the **Add Operation** command from the **Flowsheet** menu. The UnitOps view appears.
2. From the list of available unit operations, click the operation you want to install.
3. Click the **Add** button. The operation is added to the PFD. The operation property view automatically opens.

The following sections show steps for providing the minimum amount of information required to define each object. For more information, refer to the UniSim Design Operations Guide.

8.2 Defining Objects

8.2.1 Defining a Material Stream

If you specify a vapour fraction of 0 or 1, the stream is assumed to be at the bubble point or dew point. You can also specify vapour fractions between 0 and 1.

1. Add a material stream to your simulation.
2. Click the **Worksheet** tab, then select the **Conditions** page.
3. Specify values for three of the properties in the table. (One of the specifications must be temperature or pressure.)
4. Select the **Composition** page.
5. Click the **Edit** button. The Input Composition for Stream view appears.
6. Click the radio button in the Composition Basis group that indicates the basis of your composition.

7. In the table, specify the composition of your stream.
8. Click the **Normalize** button to ensure that your composition adds up to 1.0 in the case of fractions.
9. Click **Ok**.

The status bar at the bottom of the property view turns green and displays the message 'Ok'.

8.2.2 Defining an Energy Stream

1. Add an energy stream to your simulation.
2. Click the **Stream** tab.
3. In the **Heat Flow** field, specify a value for heat flow.

The status bar at the bottom of the property view turns green and displays the message 'Ok'.

8.2.3 Defining a Separator

1. Add a separator to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlets list, click the <<stream>> field and a drop-down list appears.
4. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the name of the stream.
5. Repeat step 4 if you have multiple feed streams.
6. In the Vapour Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the name of the stream.
7. In the Liquid Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the name of the stream.

When all of the attached streams are properly defined the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the separator cannot solve, the status bar is yellow and displays the requirements needed to solve.

8. (Optional) Click the **Parameters** page and enter a pressure difference in the Delta P field. (A default value of zero is entered for new separators, so it solves without entering a value, but you may not get the results you want.)

8.2.4 Defining a 3-Phase Separator

1. Add a 3-phase separator to your simulation.
 2. Click the **Design** tab, then select the **Connections** page.
 3. In the Inlets list, click the <>stream>> field and a drop-down list appears.
 4. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
 5. Repeat step 4 if you have multiple feed streams.
 6. In the Vapour drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
 7. In the Light Liquid drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
 8. In the Heavy Liquid drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
- When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the 3-phase separator cannot solve, the status bar is yellow and displays the requirements needed to solve.
9. (Optional) Click the **Parameters** page and enter a pressure difference in the Delta P field. A default value of zero is entered for new separators so it solves without entering a value, but you may not get the results you want.

8.2.5 Defining a Tank

1. Add a tank to your simulation.
 2. Click the **Design** tab, then select the **Connections** page.
 3. In the Inlets list, click the <>stream>> field to display a drop-down list.
 4. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
 5. Repeat step 4 if you have multiple feed streams.
 6. In the Vapour Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
 7. In the Liquid Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
- When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the tank cannot solve, the status bar is yellow and displays the requirements needed to solve.

8. (Optional) Click the **Parameters** page and enter a pressure difference in the Delta P field. A default value of zero is entered for new separators so it solves without entering a value, but you may not get the results you want.

8.2.6 Defining a Single Outlet Vessel

1. Add a Single Outlet Vessel to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlets list, click the <>stream>> field to display a drop-down list.
4. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. Repeat step 4 if you have multiple feed streams.
6. In the Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name. When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the Single Outlet Vessel cannot solve, the status bar is yellow and displays the requirements needed to solve.
7. (Optional) Click the **Parameters** page and enter a pressure difference in the Delta P field. A default value of zero is entered for new separators so it solves without entering a value, but you may not get the results you want.

8.2.7 Defining a Cooler/Heater

1. Add a cooler or heater to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. In the Energy drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
For your cooler/heater to solve, you must have enough specifications to calculate the pressure drop across the cooler/heater and the duty of the cooler/heater.
6. Click the **Parameters** page and enter a pressure difference for the cooler/heater in the **Delta P** field. The value is automatically calculated if the pressure is specified for both inlet and outlet streams.

7. In the **Duty** field, specify a duty for the cooler/heater. The value is automatically calculated if the temperature is specified for both the inlet and outlet streams.

When the cooler/heater is solved, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the cooler/heater cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.8 Defining an LNG Exchanger

1. Add an LNG exchanger to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlet Streams column, click the <<stream>> field and a drop-down list appears.
4. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. Repeat step 4 for the remaining inlet streams.
6. In the Outlet Streams column, click the <<stream>> field and a drop-down list appears.
7. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
8. Repeat step 7 for the remaining outlet streams.
9. In the Pressure drop column, specify a pressure drop for both the hot side and cold side.
10. Click the **Parameters (SS)** page.
11. Specify the rating method being used; heat leak/loss, pass intervals, step type or pressure profile.
For the LNG exchanger to solve, the number of independent unknowns must be equal to the number of constraints (Degrees of Freedom = 0). The LNG considers constraints to be parameters such as UA, Minimum Temperature Approach, or a temperature difference between two streams.
12. Click the **Specs (SS)** page.
13. Click the **Add** button to display the Spec view. In this view you can add a specification to define your LNG exchanger.
14. Repeat step 13 until the degrees of freedom equal zero.

When the degrees of freedom equal zero and all attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the LNG exchanger cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.9 Defining a Heat Exchanger

1. Add a heat exchanger to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Tube Side Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Tube Side Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. In the Shell Side Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
6. In the Shell Side Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
7. Click the **Parameters** page.
8. Where applicable, specify the rating method being used; heat leak/loss, heat curve details, tube and shell side pressure differences and UA.

For the heat exchanger to solve, the number of independent unknowns must equal the number of constraints (Degrees of Freedom = 0). The heat exchanger considers constraints to be parameters such as UA, Minimum Temperature Approach, or a temperature difference between two streams.
9. Click the **Specs** page.
10. Click the **Add** button to display the Spec view. In this view, add a specification to define your heat exchanger.
11. Repeat step 10 until degrees of freedom equal zero.

When the degrees of freedom equal zero and all attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the heat exchanger cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.10 Defining an Air Cooler

For information about the UniSim CFE tab in the Air Cooler view, refer to [Section 4.1 - Air Cooler](#) in the [UniSim Design Operations Guide](#).

1. Add an air cooler to your simulation.
2. Click the **Connections** page of the **Design** tab.
3. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.

For your air cooler to solve, you must have enough specifications to calculate the pressure drop across the air cooler and the duty of the air cooler.

5. Click the **Parameters** page.
6. In the **Delta P** field, specify a pressure difference for the air cooler. This value automatically calculates if the pressure is specified for both the inlet and outlet streams.
7. Specify either the overall UA **or** the temperature of the outlet air. This value automatically calculates if the temperature is specified for both the inlet and outlet streams.

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the air cooler cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.11 Defining a Pump

1. Add a pump to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. In the Energy drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
For your pump to solve, you must have enough specifications to calculate the pressure drop across the pump and the duty of the pump.
6. Click the **Parameters** page. Do one of the following:
 - In the Delta P field, specify a pressure difference for the pump. This value is calculated if the pressure is specified for both the inlet and outlet streams.
 - In the Duty field, specify the duty for the pump. These values are calculated if the pressure is specified for both the inlet and outlet streams.

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the pump cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.12 Defining a Compressor/ Expander

1. Add a compressor/expander to your simulation.
2. Click the **Design** tab, then select the **Connections** page.

3. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. In the Energy drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
For your compressor/expander to solve, you must have enough specifications to calculate the duty of the compressor/expander.
6. Select the **Parameters** page.
7. In the **Duty** field, specify a duty for the compressor/expander. This value is calculated if the pressure is specified for both the inlet and outlet streams.

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the compressor/expander cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.13 Defining a Compressible Gas Pipe

1. Add a compressible gas pipe to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. Click the **Parameters** page.
6. In the **Delta P** field, specify a pressure difference for the compressible gas pipe. By default a value of 25 kPa is specified.
7. Click the **Rating** tab, then select the **Sizing** page.
8. Specify the following parameters:
 - Length
 - Elevation Change
 - Cells
 - Roughness
 - External Diameter
 - Internal Diameter
9. Select the **Heat Transfer** page, then specify the ambient temperature and overall heat transfer coefficient in the corresponding fields.

Click the **Add Section** button to add multiple pipe sections.

When all of the attached streams are properly defined, the status bar at

the bottom of the property view turns green and displays the message 'Ok'. If the compressible gas pipe cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.14 Defining a Pipe Segment

1. Add a pipe segment to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. In the Energy drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
6. Select the **Parameters** page.
7. From the list of available pipe flow correlations, select the correlation you want to use.
8. In the **Delta P** field, specify a pressure difference for the pipe segment. This value is calculated if the pressure is specified for both the inlet and outlet streams.
9. Click the **Ratings** tab, then select the **Sizing** page.
10. Click the **Append Segment** button to display a pipe segment in the table. Specify the Fitting/Pipe, Length, and Increments.
11. Click the **View Segment** button to display the Pipe Info view. Specify the pipe schedule and diameters.
12. Click the **Heat Transfer** page.
13. In the Specify By group, click the radio button that describes how the heat transfer occurs in the pipe segment. Then specify the corresponding parameters.

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message **Ok**. If the pipe segment cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.15 Defining a Valve

1. Add a valve to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.

5. Select the **Parameters** page.
6. Specify a pressure difference for the pipe segment in the **Delta P** field. This value is calculated if the pressure is specified for both the inlet and outlet streams.

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the valve cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.16 Defining a Relief Valve

Both the inlet and outlet stream pressures must be specified for the relief valve.

1. Add a relief valve to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. Select the **Parameters** page.
6. In the **Set Pressure** field, specify the pressure for when the relief valve begins to open.
7. In the **Full Open Pressure** field, specify the pressure for when the relief valve is fully open.

When all of the attached streams are properly defined, the status bar at the bottom of the property view is either red or yellow. When the status bar is yellow, the message 'Valve is Open' displays. When the status bar is red, the message 'material flows into a closed relief valve' displays. This indicates the valve is shut and no material is passing through.

8.2.17 Defining a Mixer

1. Add a mixer to your simulation.
2. Click the **Connections** page of the **Design** tab.
3. In the Inlets list, click the <<stream>> field and a drop-down list appears.
4. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. Repeat step 4 for the multiple feed streams.
6. In the Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message

'Ok'. If the mixer cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.18 Defining a Tee

1. Add a tee to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Outlets list, click the <<stream>> field and a drop-down list appears.
5. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
6. Repeat step 5 for the multiple outlet streams.
7. Select the **Parameters** page and specify flow ratios for outlet streams.

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the tee cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.19 Defining a Simple Solid Separator

1. Add a simple solid separator to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Vapour Product drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. In the Liquid Product drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
6. In the Solids Product drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
7. Select the **Parameters** page.
8. In the **Delta P** field, specify a pressure difference for the solids separator. By default a value of 0.0 kPa is specified.
9. Select the **Splits** page, then define your stream split according to the separation.

When all of the attached streams are properly defined, the status bar at

the bottom of the property view turns green and displays the message 'Ok'. If the solid separator cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.20 Defining a Cyclone

1. Add a cyclone to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Vapour Product drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. In the Solid Product drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
6. Select the **Parameters** page.
7. In the **Particle Efficiency** field, specify a value for the particle efficiency of the cyclone.
8. Select the **Solids** page.
9. Click the **Solid Name** field to open the drop-down list. Select the solid component being separated from the stream.

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the cyclone cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.21 Defining a Hydrocyclone

1. Add a hydrocyclone to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Liquid Product drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. In the Solid Product drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
6. Select the **Parameters** page.
7. In the **Particle Efficiency** field, specify a value for the particle efficiency of the cyclone.
8. Select the **Solids** page.
9. Click the **Solid Name** field to open the drop-down list. Select the solid component being separated from the stream.

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the hydrocyclone cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.22 Defining a Rotary Vacuum Filter

1. Add a rotary vacuum filter to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Liquid Product drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. In the Solid Product drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
6. Click the **Rating** tab, then select the **Sizing** page.
7. In the Filter Size group, specify either the filter radius or filter width in the corresponding field.

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the rotary vacuum filter cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.23 Defining a Baghouse Filter

1. Add a baghouse filter to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Vapour Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. In the Solid Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the baghouse filter cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.24 Defining a Screen

1. Add a screen unit operation to your simulation.
2. Click the Design tab, then select the Connections page.
3. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Over Size drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. In the Under Size drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
6. If the Excess Product Streams check box is checked, the Over Size Excess (Optional) drop-down list will be active. In the Over Size Excess (Optional) drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
7. If Under Size Excess (Optional) drop-down list is active, in the Under Size Excess (Optional) drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.

When all of the attached streams are properly defined and screen pressure is entered, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the screen unit operation cannot solve, the status bar is red or yellow and displays the requirements needed to solve.

8.2.25 Defining a Conveyor

1. Add a conveyor to your simulation.
2. Click the Design tab then select the Connections page.
3. In the Feed drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Inlet Excess Product drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. In the Solid Product drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
6. In the Outlet Excess Product drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
7. Select the Parameters page. In the Pressure field, specify a pressure for the Conveyor.
8. Click on the Rating tab, select the Sizing page and define the parameters for conveyor operation

9. For dynamic modeling, click the Dynamics tab and specifying the pressure flow parameters.
10. When all of the attached streams and all required parameters are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If conveyor cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.26 Defining a Crusher

1. Add a crusher to your simulation.
2. Click the Design tab, then select the Connections page.
3. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. In the Energy (Optional) drop-down list select a pre-defined stream or click the space at the top of the list and type the stream name.
6. Select the Parameters page.
7. In the Pressure Drop field of the General Parameters group, specify a value for the pressure drop of the crusher.
8. If crushing one pseudo solid component into another pseudo solid component click the Inter-Component Crushing check box.
9. In the Parameters for Rousseau's Model group, for each solid component or solid group specify the Number of Sizes, Breakability, and Deviation.
10. If the power consumption is required, check the Calculate Power and Bond Theory check boxes.
11. Specify the BWI for each solid component or solid group in the Bond Work Index group.

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the crusher cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.27 Defining a CSTR

1. Add a CSTR to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlets list, click the <<stream>> field and a drop-down list appears.
4. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.

5. Repeat step 4 if you have multiple feed streams.
6. In the Vapour Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
7. In the Liquid Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
8. In the Energy drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
9. Select the **Parameters** page.
10. In the **Volume** field, specify the volume of the reactor.
11. Click the **Reactions** tab, then select the **Details** page.
12. From the Reaction Set drop-down list, select the reaction set being used.
13. From the Reaction drop-down list, select the reaction being used.

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the CSTR cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.28 Defining a PFR

1. Add a PFR to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlets list, click the <>stream>> field and a drop-down list appears.
4. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. Repeat step 4 if you have multiple feed streams.
6. In the Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
7. Select the **Parameters** page.
8. In the **Delta P** field, specify the pressure drop across the reactor.
9. Click the **Reactions** tab, then select the **Overall** page.
10. From the Reaction Set drop-down list, select the reaction set being used.
11. Select the **Details** page.
12. From the Reaction drop-down list, select the reaction being used.
13. Click the **Rating** tab, then select the **Sizing** page.
14. In the Tube Dimensions group, specify two of the three following parameters:
 - Total Volume
 - Length

- Diameter

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the PFR cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.29 Defining a Gibbs Reactor

1. Add a Gibbs reactor to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlets list, click the <<stream>> field and a drop-down list appears.
4. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. Repeat step 4 if you have multiple feed streams.
6. In the Vapour Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
7. In the Liquid Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the Gibbs reactor cannot solve, the status bar is yellow and displays the requirements needed to solve.

8. (Optional) Select the **Parameters** page and enter a pressure difference in the **Delta P** field. A default value of zero is entered for new Gibbs reactors, so your Gibbs reactor solves without entering the value, but you do not get the desired results.
9. (Optional) Click the **Reactions** tab, then select the **Overall** page. In the Reactor type group, click the corresponding radio button for the type of reactor you want to model.

8.2.30 Defining an Equilibrium Reactor

1. Add an equilibrium reactor to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlets list, click the <>stream>> field and a drop-down list appears.
4. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. Repeat step 4 if you have multiple feed streams.
6. In the Vapour Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
7. In the Liquid Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
8. Click the **Reactions** tab, then select the **Details** page.
9. From the Reaction Set drop-down list, select the reaction set containing the reaction being used.
10. From the Reaction drop-down list, select the reaction being used.
When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the equilibrium reactor cannot solve, the status bar is yellow and displays the requirements needed to solve.
11. (Optional) Click the **Design** tab, then select the **Parameters** page. In the **Delta P** field, enter a pressure difference. A default value of zero is entered for new equilibrium reactors, so your equilibrium reactor solves without entering a value, but you may not get the desired results.

8.2.31 Defining a Conversion Reactor

1. Add a conversion reactor to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlets list, click the <>stream>> field and a drop-down list appears.
4. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. Repeat step 4 if you have multiple feed streams.
6. In the Vapour Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.

7. In the Liquid Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
8. Click the **Reactions** tab, then select the **Details** page.
9. From the Reaction Set drop-down list, select the reaction set containing the reaction being used.
10. From the Reaction drop-down list, select the reaction being used.
When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the conversion reactor cannot solve, the status bar is yellow and displays the requirements needed to solve.
11. (Optional) Click the **Design** tab, then select the **Parameters** page. In the **Delta P** field, enter a pressure difference. A default value of zero is entered for new conversion reactors, so your conversion reactor solves without entering a value, but you do not get the desired results.

8.2.32 Defining a Neutralizer

This operation can only be used with a fluid package that uses the electrolyte property package.

1. Add a neutralizer to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlets list, click the <>stream>> field and a drop-down list appears.
4. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. Repeat step 4 if you have multiple feed streams.
6. In the Reagent Stream drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
7. In the Vapour Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
8. In the Liquid Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the neutralizer cannot solve, the status bar is yellow and displays the requirements needed to solve.
9. (Optional) Select the **Parameters** page. In the **Delta P** field, enter a pressure difference. A default value of zero is entered for new neutralizers, so your neutralizer solves without entering a value, but you may not get the desired results.

8.2.33 Defining a Crystallizer

This operation can only be used with a fluid package that uses the electrolyte property package.

1. Add a crystallizer to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlets list, click the <<stream>> field and a drop-down list appears.
4. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. Repeat step 4 if you have multiple feed streams.
6. In the Vapour Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
7. In the Liquid Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the crystallizer cannot solve, the status bar is yellow and displays the requirements needed to solve.
8. (Optional) Select the **Parameters** page. In the **Delta P** field, enter a pressure difference. A default value of zero is entered for new crystallizers, so your crystallizer solves without having to enter a value, but you may not get the desired results.

8.2.34 Defining a Precipitator

This operation can only be used with a fluid package that uses the electrolyte property package.

1. Add a precipitator to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlets list, click the <<stream>> field and a drop-down list appears.
4. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. Repeat step 4 if you have multiple feed streams.
6. In the Reagent Stream drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
7. In the Vapour Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
8. In the Liquid Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the precipitator cannot solve, the status bar is yellow and displays the requirements needed to solve.

9. (Optional) Select the **Parameters** page. In the **Delta P** field, enter a pressure difference. A default value of zero is entered for new precipitators, so your precipitator solves without entering a value, but you may not get the desired results.

8.2.35 Defining a Distillation Column

Use this procedure to define a distillation column using the input expert:

1. Add a distillation column to your simulation.
2. In the **# Stages** field, specify the number of trays in the column.
3. In the Inlet Streams list, click the <> stream > field and a drop-down list appears.
4. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. Repeat step 4 if you have multiple feed streams.
6. In the Inlet Stage column, use the drop-down list to select what stage the stream is entering the column.
7. In the Condenser Energy Stream drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
8. In the Condenser group, click the condenser type radio button that you want to use for the column. Depending on the selection, select a pre-defined stream or click the space at the top of the list and type the stream name for the following:
 - **Total**. Specify in the Ovhd Liquid Outlet drop-down list.
 - **Partial**. Specify in the Ovhd Outlets drop-down list.
 - **Full Rflx**. Specify in the Ovhd Vapour Outlet drop-down list.
9. In the Reboiler Energy stream drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
10. In the Bottoms Liquid Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
11. Click the **Next** button.
12. In the **Condenser Pressure** field, enter the pressure in the condenser.
13. In the **Condenser Pressure Drop** field, enter the pressure drop across the condenser.
14. In the **Reboiler Pressure** field, enter the pressure in the reboiler.
15. Click the **Next** button.

16. (Optional) Enter temperature values for the condenser, top stage, and reboiler.

17. Click the **Next** button.

18. (Optional) Enter a product flow rate and a reflux ratio.

19. Click **Done** to display the Column property view.

For the distillation column to solve, the number of independent unknowns must be equal to the number of constraints (Degrees of Freedom = 0). The distillation column considers constraints to be parameters such as product draw rates, tray temperatures, or reflux ratio.

20. Click the **Design** tab, then select the **Specs** page.

21. Click the **Add** button to display the Spec view. In this view, add a specification to define your distillation column. Repeat this step until the degrees of freedom equal zero.

When the degrees of freedom equal zero and all attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the distillation column cannot solve, the status bar is red and displays 'Unconverged'.

8.2.36 Defining a Refluxed Absorber Column

Use this procedure to define a refluxed absorber column using the input expert:

1. Add a refluxed absorber column to your simulation.
2. In the **# Stages** field, type the number of trays for the column.
3. In the Bottom Stage Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Condenser Energy Stream drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. In the Condenser group, click the condenser type radio button being used for the column. Depending on the selection, in the Ovhd Liquid Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream names for the following:
 - Total
 - Partial
 - Full Rflx
6. In the Bottoms Liquid Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.

7. Click the **Next** button.
8. In the **Condenser Pressure** field, enter the pressure in the condenser.
9. In the **Condenser Pressure Drop** field, enter the pressure drop across the condenser.
10. In the **Reboiler Pressure** field, enter the pressure in the reboiler.
11. Click the **Next** button.
12. (Optional) Enter temperature values for the condenser, top stage, and reboiler.
13. Click the **Next** button.
14. (Optional) Enter the product flow rate and a reflux ratio.
15. Click **Done** to display the Column property view.

For the refluxed absorber column to solve, the number of independent unknowns must equal the number of constraints (Degrees of Freedom = 0). The refluxed absorber column views constraints as parameters such as product draw rates, tray temperatures, or reflux ratio.
16. Click the **Design** tab, then select the **Specs** page.
17. Click the **Add** button to display the Spec view and add a specification to define your refluxed absorber column.
18. Repeat step 17 until the degrees of freedom equal zero.

Select the Monitor page of the Design tab to see how many degrees of freedom the column has and manage the specifications in the column.

When the degrees of freedom equal zero and all attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the refluxed absorber column cannot solve, the status bar is red and displays 'Unconverged'.

8.2.37 Defining an Absorber Column

Use this procedure to define an absorber column using the input expert:

1. Add an absorber column to your simulation.
2. In the **# Stages** field, type the number of trays in the column.
3. In the Top Stg Reflux group, click the radio button being used for the column. Depending on the selection, select a pre-defined stream or click the space at the top of the list and type the stream names for the following:
 - **Liquid Inlet**. Specify in the Top Stage Inlet drop-down list.
 - **Pump-around**. Specify in the Draw Stage drop-down list.
4. In the Bottom Stage Inlet column, use the drop-down list to specify at what stage the stream enters the column.

5. In the Ovhd Vapour Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
6. In the Bottoms Liquid Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
7. Click the **Next** button.
8. In the **Top Stage Pressure** field, type the pressure in the condenser.
9. In the **Bottom Stage Pressure** field, enter the pressure in the reboiler.
10. Click the **Next** button.
11. (Optional) Enter temperature values for the top stage and bottom stage.
12. Click **Done** to display the Column property view.

When all attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the absorber column cannot solve, the status bar is red and displays 'Unconverged'.

8.2.38 Defining a Reboiled Absorber Column

Use this procedure to define a reboiled absorber column using the input expert:

1. Add a reboiled absorber column to your simulation.
2. In the **# Stages** field, specify the number of trays in the column.
3. In the Top Stg Reflux group, click the radio button being used for the column. Depending on the selection, select a pre-defined stream or click the space at the top of the list and type the stream name for the following:
 - **Liquid Inlet**. Specify in the Top Stage Inlet drop-down list.
 - **Pump-around**. Specify in the Draw Stage drop-down list.
4. In the Ovhd Vapour Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. In the Reboiler Energy Stream drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
6. In the Bottoms Liquid Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.

7. Click the **Next** button.
8. In the **Top Stage Pressure** field, type the pressure in the condenser.
9. In the **Reboiler Pressure** field, type the pressure in the reboiler.
10. Click the **Next** button.
11. (Optional) Enter temperature values for the top stage and reboiler.
12. Click the **Next** button.
13. (Optional) Enter a boilup ratio.
14. Click **Done** to display the Column property view.

For the reboiled absorber column to solve, the number of independent unknowns must equal the number of constraints (Degrees of Freedom = 0). The reboiled absorber column views constraints as parameters such as product draw rates, tray temperatures, or reflux ratio.

15. Click the **Design** tab, then select the **Specs** page.
16. Click the **Add** button to display the Spec view. From this view, add a specification to define your refluxed absorber column.
17. Repeat step 16 until degrees of freedom equals zero.

When the degrees of freedom equal zero and attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the reboiled absorber column cannot solve, the status bar is red and displays 'Unconverged'.

8.2.39 Defining a Liquid-Liquid Extractor

Use this procedure to define a liquid-liquid extractor using the input expert:

1. Add a liquid-liquid extractor to your simulation.
2. In the **# Stages** field, specify the number of trays in the column.
3. In the Top Stage Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Bottom Stage Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. In the Ovhd Light Liquid drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
6. In the Bottoms Heavy Liquid drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.

Click the Monitor page of the Design tab to see how many degrees of freedom the column has and manage the specifications in the column.

7. Click the **Next** button.
8. In the **Top Stage Pressure** field, enter the pressure in the condenser.
9. In the **Bottom Stage Pressure** field, enter the pressure in the reboiler.
10. Click the **Next** button.
11. (Optional) Enter temperature values for the top stage and bottom stage.
12. Click **Done** to display the Column property view.

When all attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the liquid-liquid extractor cannot solve, the status bar is red and displays 'Unconverged'.

8.2.40 Defining a Three Phase Distillation Column

Use this procedure to define a three phase distillation column using the input expert:

1. Add a three phase distillation column to your simulation.
2. Click one of the following radio buttons for the column you want to model:
 - Distillation
 - Refluxed Absorber
 - Reboiled Absorber
 - Absorber
3. Click the **Next** button.
4. In the Two Liquid Phase Check group, select the stages you want to check for two liquid phases.
5. Click the **Next** button.
6. In the Condenser Energy Stream drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
7. Use the radio buttons in the Condenser type, Outlet Streams, and Reflux Streams groups to specify the condenser configuration of the column.
8. In the remaining drop-down lists, select a pre-defined stream or click the space at the top of the list and type the stream name.
9. Click the **Next** button.
10. (Optional) Specify the flow rates for entering and leaving the condenser.

11. Click the **Next** button to display the Input Expert view for the selected column.
12. Follow the sections outlined for each of the columns above to complete the three phase distillation column.

8.2.41 Defining a Component Splitter

1. Add a component splitter to your simulation.
2. In the Inlet Streams list, select a pre-defined stream from the drop-down list, or click the space at the top of the list and type the stream name.
3. Repeat step 2 if you have multiple feed streams.
4. In the Energy Streams list, select a pre-defined stream from the drop-down list, or click the space at the top of the list and type the stream name.
5. Repeat step 4 if you have multiple energy streams.
6. In the Overhead Outlet list, click the <<stream>> field to display the drop-down list. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
7. Repeat step 6 if you have multiple outlet streams.
8. In the Bottoms Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
9. Select the **Splits** page.
10. Specify the split fraction for each of the overhead streams.

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the component splitter cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.42 Defining a Short Cut Distillation Column

1. Add a short cut distillation column to your simulation.
2. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
3. From the Top Product Phase group, select whether you want the overhead product to be vapour or liquid.
4. Then specify the following by selecting a pre-defined stream or clicking the space at the top of the list and typing the stream name:

- **Liquid**. Specify in the Distillate drop-down list.
 - **Vapour**. Specify in the Overhead Vapour drop-down list.
5. In the Condenser Duty drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
 6. In the Reboiler Duty drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
 7. In the Bottoms drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
 8. Select the **Parameters** page.
 9. Click the down arrow  in the Light Key in Bottoms cell under the Component column. From the drop-down list of available components, select the component to use for the light key.
 10. In the Light Key in Bottoms cell under the Mole Fraction column, specify the mole fraction of the key component.
 11. Click the down arrow  in the Heavy Key in Distillate cell under the Component column. From the list of available components, select the component to use for the heavy key.
 12. In the Heavy Key in Distillate cell under the Mole Fraction column, specify the mole fraction of the key component.
 13. In the **Condenser Pressure** field, specify the pressure at the condenser.
 14. In the **Reboiler Pressure** field, specify the pressure at the reboiler.
 15. In the **External Reflux Ratio** field, specify the external reflux ratio.

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the short cut distillation column cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.43 Defining an Adjust

1. Add an adjust to your simulation.
2. In the Adjusted Variable group, click the **Select Var** button to display the Select Variable view.
3. Select the object and the variable you want to use and click **OK**.
4. In the Target Variable group, click the **Select Var** button to display the Select Variable view.
5. Select the object and the variable you want to use and click **OK**.
6. In the Target Value group, enter either a user specified value or a value from another object.
7. Click the **Start** button to start the solver.

8.2.44 Defining a Set

1. Add a set to your simulation.
2. In the Target Variable group, click the **Select Var** button to display the Select Variable view.
3. Select the object and the variable you want to use and click **OK**.
4. In the Source Object drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. Click the **Parameters** tab.
6. Specify either the multiplier or offset in the appropriate field. The parameter that is not specified is automatically calculated.

8.2.45 Defining a Recycle

1. Add a recycle to your simulation.
2. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
3. In the Outlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.

8.2.46 Defining a PID Controller

1. Add a PID controller to your simulation.
2. In the Process Variable Source group, click the **Select Var** button to display the Select Variable view.
3. Select the object and the variable you want to use and click **OK**.
4. In the Output Target Object group, click the **Select OP** button to display the Select OP Object view.
5. Select the object you want to use and click **OK**.
6. Click the **Parameters** tab, then select the **Configuration** page.
7. Click either the **Reverse** or **Direct action** radio button.
8. In the **Kc** field, specify a value for the controller gain.
9. In the **Ti** field, specify a value for the integral time, if required.
10. In the **Td** field, specify a value for the derivative time, if required.
11. In the **PV Minimum** field, specify the minimum process variable value.
12. In the **PV Maximum** field, specify the maximum process variable value.
13. From the mode drop-down list, select one of the following options for the mode of the controller:
 - Man

- Auto Indicator
- Off

14. In the **SP** field, specify the set point for the controller.

8.2.47 Defining a Selector Block

1. Add a selector block to your simulation.
2. Click the **Add PV** button to display the Select Input PV view.
3. Select the object and the variable you want to use and click **OK**.
4. Repeat steps #2 and #3 for each process variable you want to use.
5. Click the **Select PV** button to display the Select Input PV view.
6. Select the object and the variable you want to use and click **OK**.

8.2.48 Defining a Balance

1. Add a balance to your simulation.
2. Click the **Connections** tab.
3. In the Inlet Streams list, click the <<stream>> field to display a drop-down list. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. Repeat step 3 if you have multiple feed streams.
5. In the Outlet Streams list, click the <<stream>> field to display a drop-down list. From the drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
6. Repeat step 5 if you have multiple outlet streams.
7. Click the **Parameters** tab.
8. In the Balance Type group, select the balance type you want to use.

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the balance cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.49 Defining a Digital Point

1. Add a digital point to your simulation.
2. In the Output Target Object group, click the **Select OP** button to display the Select OP Object view.
3. Select the object you want to use.
4. Click **OK**.

8.2.50 Defining a Transfer Function Block

1. Add a transfer function block to your simulation.
2. In the Transformed PV Target group, click the **Select PV** button to display the Select Output PV view.
3. Select the object and the variable you want to use.
4. Click **OK**.
5. Click the **Parameters** tab, then select the **Configuration** page.
6. Specify a value for the input variable in the **Constant PV** field.

8.2.51 Defining an MPC Controller

1. Add an MPC controller to your simulation.
2. In the Process Variable Source group, click the **Select PV** button to display the Select Input PV view.
3. Select the object and the variable you want to use.
4. Click **OK**.
5. In the Output Target Object group, click the **Select OP** button to display the Select OP Object view.
6. Select the object and the variable you want to use and click **OK**.
7. Click the **Control Valve** button to display the Control Valve view.
8. Depending on what type of stream the output object is, specify the minimum and maximum variables and then close the view.
9. Click the **Parameters** tab, then select the **Configuration** page.
10. In the PV Min and Max group, specify minimum and maximum values for the process variables.

8.2.52 Defining a Centrifuge

1. Add a centrifuge to your simulation.
2. Click the **Design** tab, then select the **Connections** page.
3. In the Inlet drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the Liquid Product drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. In the Solid Product drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.

6. In the Energy drop down list, if you leave it unattached to an energy stream. The centrifuge will operate at adiabatic condition. If you choose to add a pre-defined Energy stream or click the space at the top of the list and type the Energy stream name, you have to either specify a duty or the centrifuge operation temperature.
7. Select the **Parameters** page.
8. In the **Model Selection** drop down list, select either Laval (Sediment) or Bird (Sediment) model.
9. In the field of the **General Parameters** group, specify a value either for the **pressure drop** or **operating pressure** of the centrifuge. Specify **operating temperature** if Energy stream is attached to the centrifuge and duty is not specified.
10. In the **Component Separation Efficiency** group, for each solid, specify the efficiency. A default 100% is provided.
11. Select the Split page.
12. In the **Split Weight Fraction** group, specify a value for **Feed Liquid to Solid Product Stream** and **Feed Solid to Solid Product Stream**.
13. Click the **Rating** tab, then select the **Sizing** page.
14. Specify the sizing values for the centrifuge. For Laval (Sediment) model, specify **number of stacks**, **Disc Stack Slant Length**, **Inner Disc Stack Radius**, **Outer Disc Stack Radius**, **Disc Stack Slant Angle** and **Rotational Speed**. For Bird (Sediment) model, specify **Bowl Radius**, **Bowl Length**, **Weir Height** and **Rational Speed**.

When all of the attached streams are properly defined, the status bar at the bottom of the property view turns green and displays the message 'Ok'. If the centrifuge cannot solve, the status bar is yellow and displays the requirements needed to solve.

8.2.53 Defining a Virtual Stream

1. Add a virtual stream to your simulation.
2. Click **Connections** Tab.
3. In the **Reference Stream** drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
4. In the **Target Stream** drop-down list, select a pre-defined stream or click the space at the top of the list and type the stream name.
5. Click the **Parameters** tab. Transfer Information page will open.
6. The **Multiplier** and the **offset** have default values at 1 and 0. The User can modify it as needed.

When both Reference and target streams are solved according to the

transfer parameters, the status bar at the bottom of the property view turns green and displays the message 'Ok', which means the transformation is completed. If the Virtual stream cannot solve, the status bar will display the required information needed to solve with yellow color.

9 Print Options

9.1 Introduction	2
9.2 Printing in UniSim Design	2
9.2.1 Menu Bar Options	3
9.2.2 Printing Datasheets	3
9.2.3 Printing the PFD.....	5
9.2.4 Printing Plots.....	5
9.2.5 Printer Setup.....	6
9.3 Reports.....	7
9.3.1 Creating a Report.....	8
9.3.2 Editing a Report.....	11
9.3.3 Deleting a Report.....	11
9.3.4 Report Format & Layout	11
9.3.5 Text Report Format	12
9.3.6 Specs Only.....	13
9.3.7 Printing & Previewing Reports	14
9.4 Printing the PFD as a File.....	14

9.1 Introduction

In UniSim Design, you can transcribe process information concerning your simulation case using printing features. You can create printed reports ranging from basic data to comprehensive summaries. There are two primary printing options in UniSim Design:

- Object Specific
- Reports

Object Specific printing relates to the current object of focus in the simulation case. For instance, if the Separator property view is active, you can print a Snapshot of that view as seen on your monitor or print out a Datasheet specific to the unit operation. A Datasheet displays object related information that can include input specifications and calculated results. Each object within UniSim Design has at least one Datasheet available, with many objects also having condensed versions of the full Datasheet.

Printing reports deal with more extensive information sets. When creating a report, you collect the Datasheets of multiple objects into one document. UniSim Design enables you to select any Datasheet for any object currently in your simulation case. For each report created, you can customize the page setup.

Both primary printing features are accessed through the menu bar, however, Object Specific printing is also available through the Object Inspect menu. By right-clicking the Title Bar of an object, you can preview and print its associated Datasheet.

The Report view can remain open while you manipulate your flowsheet. If changes occur that affect the values shown in a Datasheet, you can easily update the information.

9.2 Printing in UniSim Design

In UniSim Design, you can print information in one of three ways:

Refer to [Section 9.2.1 - Menu Bar Options](#) for details.

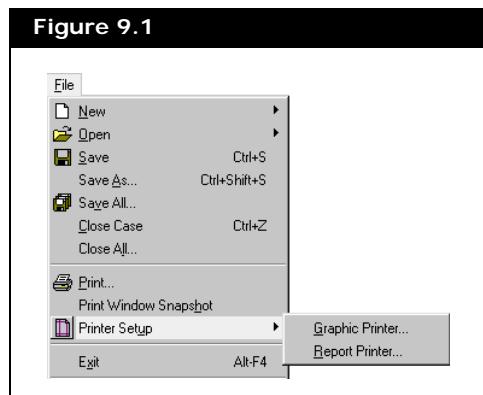
Refer to [Section 9.3 - Reports](#) for details on the Report Manager.

Method	Description
Menu Bar	Select one of the options under the File menu.
Object Inspect menu	Right-click the Title Bar of a view and select the Print Datasheet command from the menu. This is the same as selecting the Print command from the File menu.
Report Manager	In the Simulation environment, from the Tools menu, select Reports. This opens the Report Manager view.

9.2.1 Menu Bar Options

The UniSim Design printing options found in the File in the menu are as follows:

Option	Description
Print	Lists the available Datasheet(s) for the currently active object. You can select a Datasheet and either preview or print it. See Section 9.2.2 - Printing Datasheets for more detail.
Print Window Snapshot	Prints a bitmap of the currently active UniSim Design view. Use this command when you want to print a view that does not have a Datasheet associated with it (i.e., table such as a Column Profiles table).
Printer Setup	With this command, you can select either the Graphic Printer or the Report Printer . This enables you to select the printer, paper orientation, paper size and source. Refer to Section 9.3.7 - Printing & Previewing Reports for details.

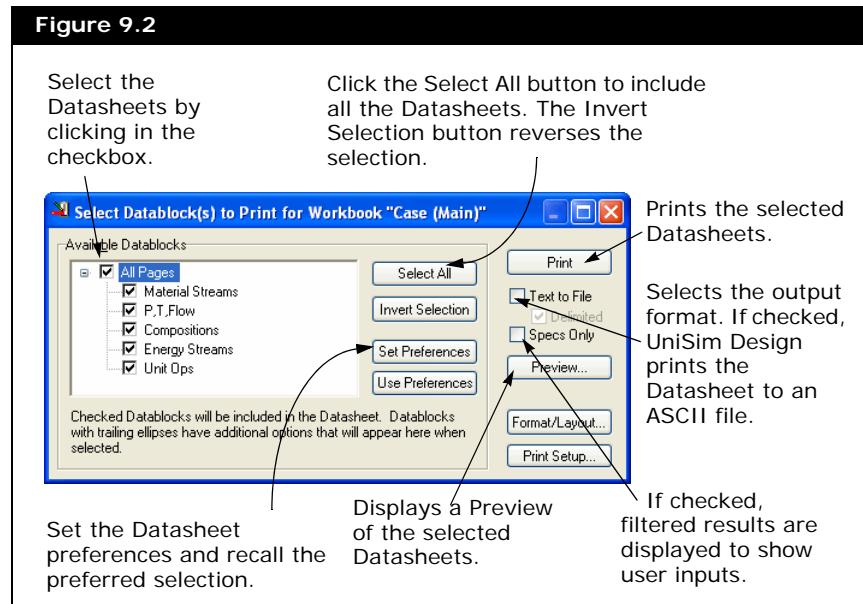


9.2.2 Printing Datasheets

All default and customized Workbook pages have Datasheets available.

When you select the Print command, the Select Datablocks to Print view appears and UniSim Design prepares to print the Datasheets for the view that currently has focus. For example, 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 available (e.g., the Reaction Package view), the Select Datablocks to Print view appears with no Datasheets available.

The Select Datablocks to Print appears as follows.



Click the Set Preferences button to save the Datasheet selections set in the UniSim Design preferences. This allows you to use the same settings for each type of object. For example, if you print the same Datasheets for every stream in your case, use the Set Preferences button to save the settings for the first stream and the Use Preferences button to load the settings into each of the remaining streams.

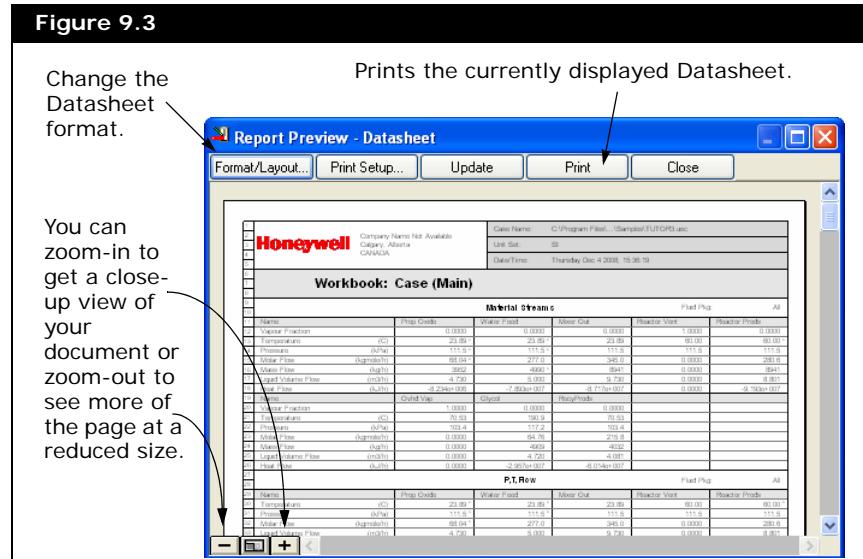
The default Datasheet selections are set in the Session Preferences. Refer to [Section 12.4 - Reports Tab](#) for more details.

There are particular Datasheets for certain operations. For example, a column tray section has a Tray Section Component Summary and a Datasheet; a Strip Chart has a Strip Chart Variables and Historical Data Datasheet.

The active location in the flowsheet determines which Datasheets are available. In this case, the active location is a Workbook, so the only available options pertain to the Workbook. The All Pages Datasheet displays all the information in the Workbook. Only the variables present on the Workbook tabs appear in the Datasheets.

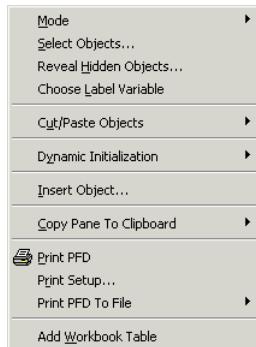
Click the Preview button to open the Report Preview view as shown

below.



9.2.3 Printing the PFD

Alternatively, you can select Print from the File menu when the PFD has focus.



To print the PFD, right-click on an empty area of the PFD to display the Object Inspect menu shown in the figure on the left.

There are three print related functions available:

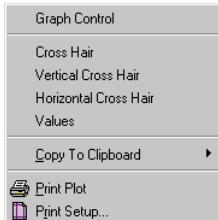
Option	Description
Print PFD	Prints the PFD as it appears on the screen. Only the sections visible within the PFD view are printed. Any tables that you add are also printed. When you select this command, the PFD is printed without accessing any further menus.
Print Setup	Accesses the Windows Print Setup. You can set the printer, the paper orientation, the paper size and paper source.
Print to PFD to File	Prints the entire PFD to a file. For more information refer to Section 9.4 - Printing the PFD as a File .

All items (Streams, Operations, Text, and PFD Tables) included in the PFD view can be printed. You can also use the Print Snapshot command under the File menu to print the PFD when it has focus.

9.2.4 Printing Plots

To print a plot, right-click on the plot area to display the Object Inspect menu as shown on the left. Select one of the following two printing

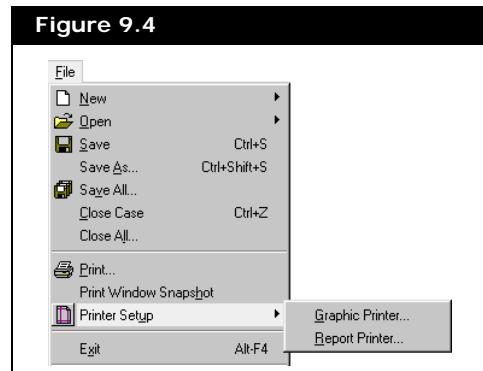
options from the menu:



Plot Object Inspect menu

9.2.5 Printer Setup

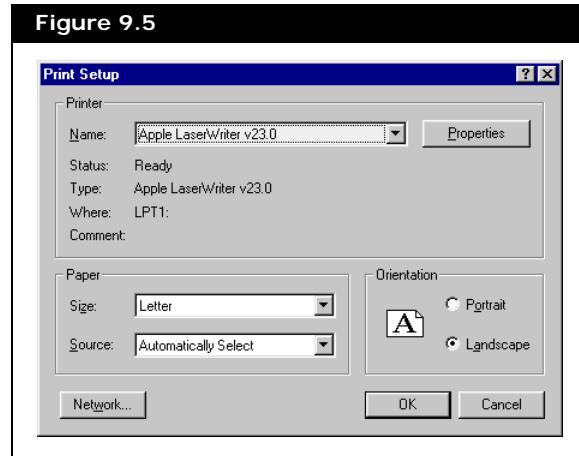
The Print Setup view is accessed by selecting Printer Setup from the File menu. The sub-menu provides two additional commands:



Printer	Description
Graphic Printer	The Graphic Printer is used to print the PFD, Plots, Strip Charts and Snapshots.
Report Printer	The Report Printer is used to print Datasheets, Reports and Text.

If the Print Setup view is accessed by right-clicking (in other words, through the PFD Object Inspect menu), UniSim Design defaults to the appropriate printer according to the active location in the flowsheet.

The layout of the Print Setup view varies depending on the selected printer. You can also modify the default properties for the selected printer by clicking the Properties button.

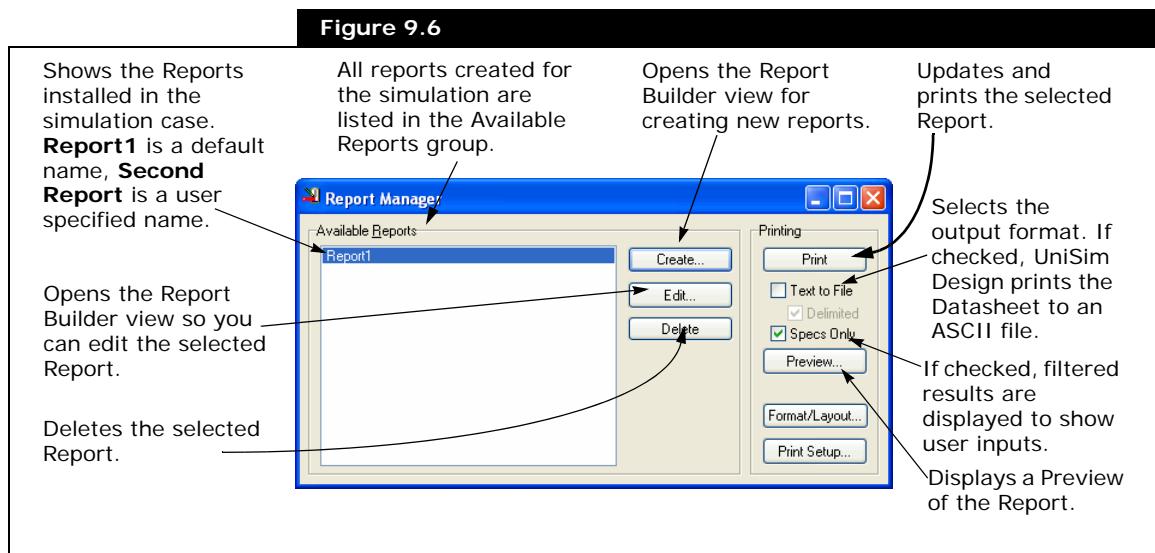


9.3 Reports

The Report Manager can only be accessed in the Simulation environment.

Within a simulation case, you can print stream and operation Datasheets using the print function, however, this only enables you to print Datasheets for a single object at a time. By using the Report Manager, you can combine multiple Datasheets for streams and operations in a single report and print the entire document. You can also format the display of the report to meet your requirements.

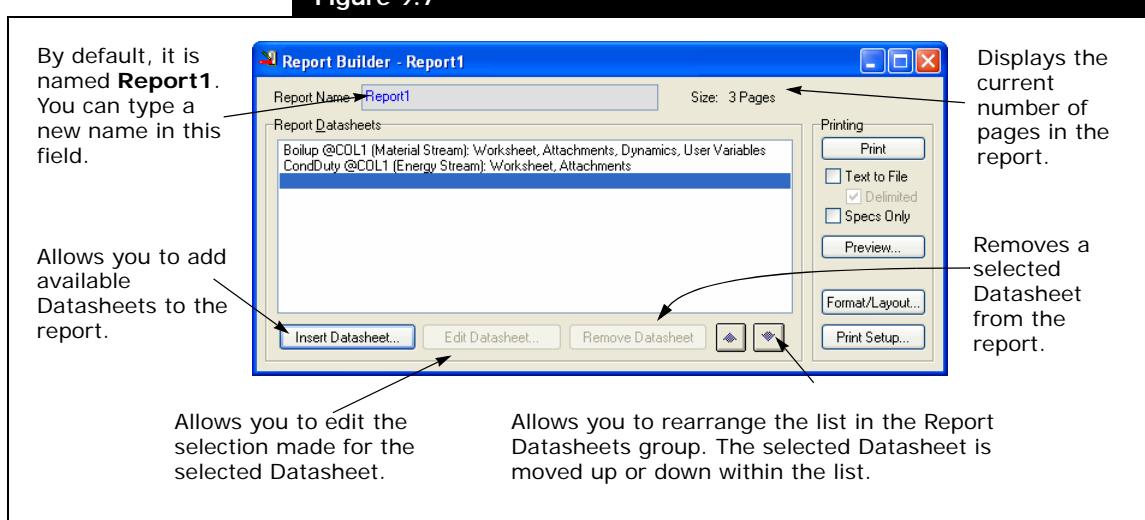
The Report Manager is accessed by selecting Reports from the Tools menu, or by using the hot key combination **CTRL R**.



9.3.1 Creating a Report

Click the Create button in the Report Manager to open the Report Builder view for creating a new report.

Figure 9.7

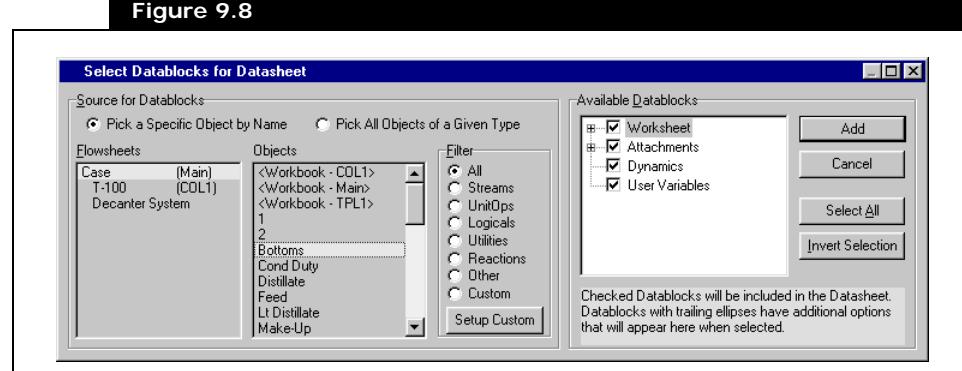


The Printing group provides the same functions as what you find in the Report Manager.

Inserting a Datasheet

Click the Insert Datasheet button in the Report Builder view to open the Select Datablocks for Datasheet view as shown below.

Figure 9.8



To return to the Report Builder view without adding a Datasheet, click the Cancel button.

The view is divided into the Source for Datablocks and Available Datablocks groups. The Source for Datablocks Group contains a list of available flowsheets and the following radio buttons:

- Pick a Specific Object by Name
- Pick All Objects of a Given Type

The information contained within the group changes depending on the radio button selected.

The Available Datablocks group lists the datablocks that can be added to the report.

Pick a Specific Object by Name

Use the Filter to narrow the search list of available objects.

Expand some of branches in the tree by clicking the + symbol to reveal more datablocks.

When the Pick a Specific Object by Name radio button is selected, the Objects list and Filter group appear as shown in [Figure 9.8](#). This allows you to insert individual Datasheets for any object that is present in the simulation case.

To add a Datasheet to your report, do the following:

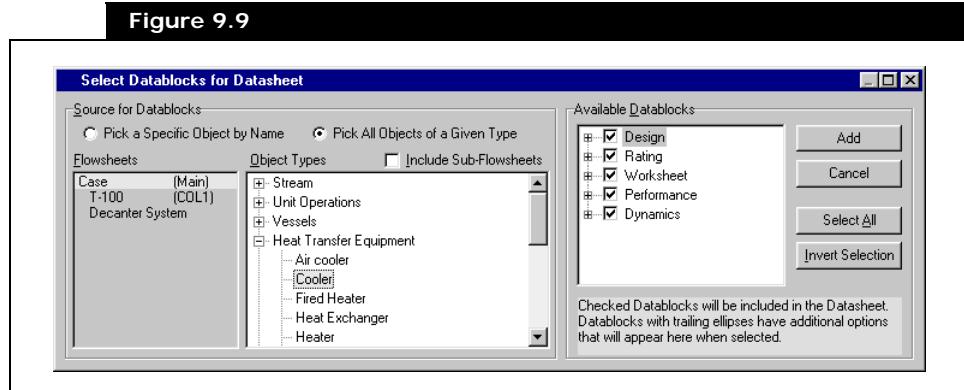
1. From the list of available flowsheets, click the flowsheet containing the objects you want to add to the Datasheet.
2. From the list of available objects, select an object.
3. In the Available Datablocks tree, all datablocks available for the selected flowsheet object appear. Select the datablocks you want to add to the report by clicking in their corresponding checkboxes.
4. Click the **Add** button to add the datablocks with checkmarks to the report.
5. Repeat steps 1 to 4 for each Datasheet that you want to add to the report.
6. Click the **Done** button when all Datasheets are added.

Pick All Objects of a Given Type

When the Pick All Objects of a Given Type radio button is selected, the list of available object types appears as shown in the figure below. This enables you to insert Datasheets for all object types in the simulation

case.

Figure 9.9



To add a group of Datasheets, do the following:

1. From the list of available flowsheets, select the flowsheet containing the objects you want to add the Datasheet.
2. In the list of available object types, select an object type (for example, material stream or compressor). To open the list of sub-items, click the +.
3. In the Available Datablocks tree, all datablocks available for the selected flowsheet object appear. Select the datablocks you want to add to the report by clicking the corresponding checkbox.
4. Click the **Add** button to add the datablocks with checkmarks to the report.
5. Repeat steps 1 to 6 for each Datasheet being added to the report.
6. When all Datasheets are added, click the **Done** button.

Editing a Datasheet

1. From the list of available report Datasheets in the Report Builder view, select the Datasheet you want to edit.
2. Click the **Edit Datasheet** button to open the Select Datablocks for Datasheet view. Refer to the section **Inserting a Datasheet** for more information about this view.
3. Use this view to edit the Datasheet.

Removing a Datasheet

1. From the list of available report Datasheet in the Report Builder view, click the Datasheet you want to delete.
2. Click the **Remove Datasheet** button. You are not prompted to confirm the deletion of the Datasheet.

For the selected object type, add Datasheets to the report for objects that reside within sub-flowsheets. Check the **Include Sub-Flowsheets** checkbox.

9.3.2 Editing a Report

1. From the list of available reports in the Report Manager view, click the report you want to edit.
2. Click the **Edit** button to open the Report Builder view. Refer to [Section 9.3.1 - Creating a Report](#) for more information about this view.

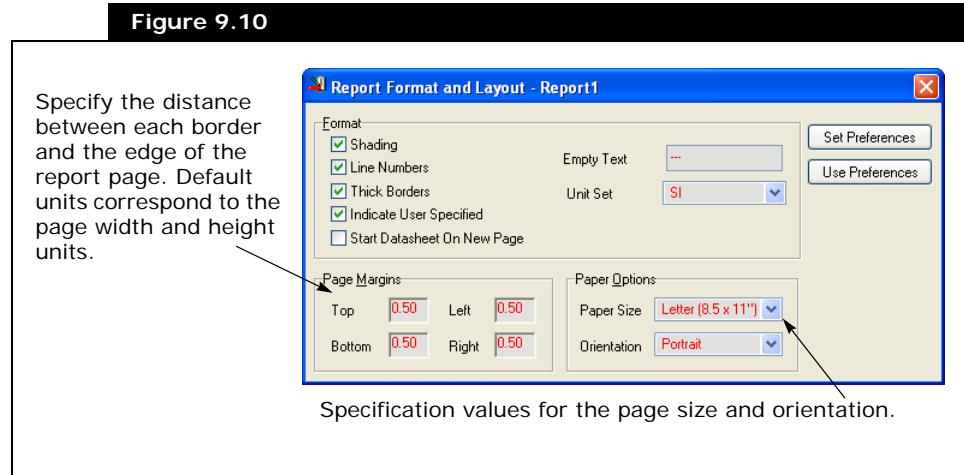
9.3.3 Deleting a Report

1. From the list of available reports in the Report Manager view, click the report you want to delete.
2. Click the **Delete** button. You are not prompted to confirm the deletion of the Datasheet.

9.3.4 Report Format & Layout

When the Text to File checkbox in the Report Builder and Report Manager views is unchecked, the **Format/Layout** button appears. Click this button to open the Report Format and Layout view. This view displays the options available for customizing the format and layout of your report.

Figure 9.10



The following options in the Format group allow you to determine how the data will appear in the report:

Field	Description
Shading	When selected, the headers, footers and titles are shaded.
Line Number	When selected, line numbers appear on the left side of each page.

Field	Description
Thick Borders	When selected, the report borders are thicker than the other lines in the report.
Indicate User Specified	When selected, any user specified values in the Datasheet are indicated with an asterisk “*”.
Start Datasheet on New Page	When selected, each Datasheet starts on a new page.
Empty Text	Specify what displays in the report when there is no value available. The default is ---.
Unit Set	Select the unit set you want your report to use. This gives you the option of printing reports with different unit sets than your case. For example, your case may be in SI, but you require your report to be in Field units.

The default report format is set in the Session Preferences. See [Section 12.4 - Reports Tab](#) for more details.

Click the Set Preferences button to save your format selections in UniSim Design preferences. This enables you to use the same settings for each report you create. Click the Use Preferences button to load the settings into any future reports.

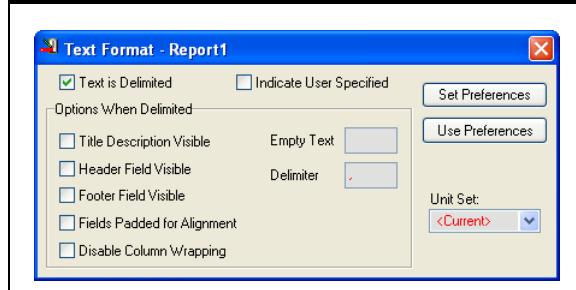
9.3.5 Text Report Format

When the **Text to File** checkbox in the Report Builder and Report Manager views is checked, the Delimited checkbox is available and the Format button appears.

Check the **Delimited** checkbox if you want your text file to be text delimited.

Click the **Format** button to open the Text Format view. This view displays the options for formatting your text file.

Figure 9.11



The items in the Options When Delimited group allow you to control

how data appears in the report:

Field	Description
Text is Delimited	When selected, the text file will be delimited.
Title Description Visible	When selected, a title is added to the text file. The title includes the name of the object and the tabs in the report.
Header Field Visible	When selected, a header is added to the text file. The header includes the company information and the date the file was created.
Footer Field Visible	When selected, a footer is added to the text file. The footer includes the UniSim Design version and build number.
Fields Padded for Alignment	When selected, spaces are added between each field to align the fields in the report.
Disable Column Wrapping	When selected, column wrapping is disabled. This means that text running past the edge of the page does not wrap onto the next line.
Empty Text	Specify what you want to display in the report when there is no value available. The default is ---.
Delimiter	Specify what you want to use as the delimiter in your text file. The UniSim Design default is '/'.

From the Unit Set drop-down list, select the unit set you want your report to use. This gives you the option of printing reports with different unit sets than in your case. For example, your case may be in SI, but you can print the report in Field units.

The default report format is set in the Session Preferences. See [Section 12.4 - Reports Tab](#) for more details.

Click the Set Preferences button to save the format selections you defined in UniSim Design preferences. This allows you to use the same settings for each report you create. Click the Use Preferences button to load the settings into subsequent reports.

9.3.6 Specs Only

The **Specs Only** feature is available when printing reports. By default it is turned off but a user can choose to select a **Specs Only** checkbox before printing a Report or Datasheet. The checkbox appears just below the **Text to File** checkbox on the datablocks selection view (i.e. just before printing). When the option is turned on for any report the results are filtered to just show user inputs. This works for all printing styles (previewing, printing or writing to a file). In some cases values will be shown that are not user input in order to provide context. The Indicate User Specified behaviour is implicitly turned on (regardless of user's choice) when **Specs Only** is selected.

9.3.7 Printing & Previewing Reports

Refer to [Section 9.2.2 - Printing Datasheets](#) for more information regarding the print preview view.

A report can be printed or previewed from either the Report Manager view or the Report Builder view. Click the Print button in either of these views and the entire report is updated and printed.

When you preview a report, a print button is available on the preview screen so you can print directly from the preview.

9.4 Printing the PFD as a File

To open the *.dxf file in AutoCAD Release 14, select DXF in the Files of Type drop-down list on the File Open view.

You can use UniSim Design to create an ASCII Drawing Interchange File (DXF) representing the UniSim Design PFD. The *.dxf file can then be read into AutoCAD. By default, the file created is called pfd.dxf and contains 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)

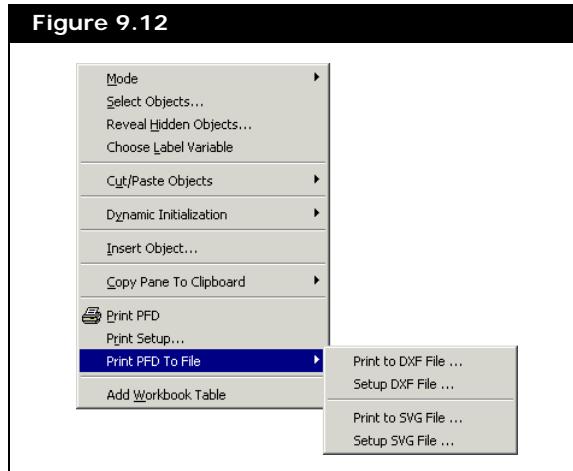
Each layer is created using standard AUTOCAD blocks.

To create a *.dxf file in UniSim Design, do the following:

1. Right-click the PFD and select **Print PFD to File** from the Object

Inspect menu.

Figure 9.12



2. Select **Print to DXF File** from the sub-menu. A standard windows file selection view appears.
3. Select the path and file name for the *.dxf file.

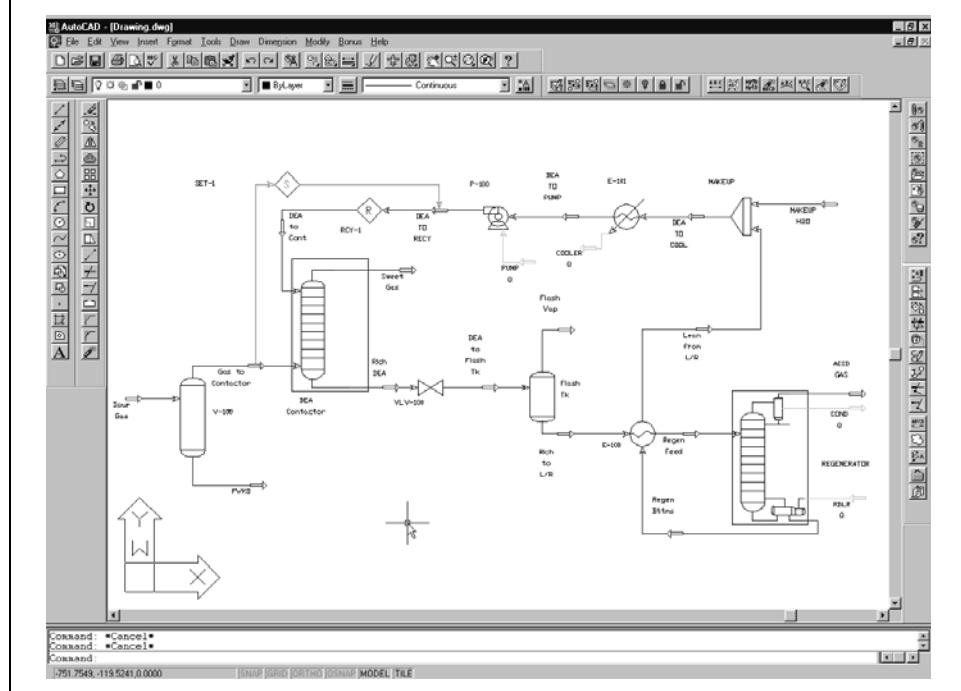
You can also right-click the PFD and select **Print PFD to File - Setup DXF File**. This opens the Setup DXF view. This view enables you to set which layers are sent to the file. Click the Print button, define the file location, and click the Save button.

Figure 9.13



An example of a pfd.dxf exported to AutoCAD appears below:

Figure 9.14



Included in the directory **\UniSim Design\Support** is a header file named autocad.hdr. If you have trouble transferring the *.dxf into AutoCAD, changes may be needed in 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.

To check the header file, follow the steps below:

1. Start a completely new AutoCAD drawing.
2. Verify that the custom corporate title blocks and border are not in the blank drawing 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 UniSim Design.

10 Edit Options

10.1 Introduction	3
10.2 Edit Menu.....	3
10.3 Editing the PFD	4
10.3.1 PFD Menu.....	4
10.3.2 Object Inspect Menu.....	5
10.3.3 PFD Tools	6
10.3.4 Selecting PFD Objects	12
10.3.5 Deselecting PFD Objects.....	13
10.3.6 Moving Objects	14
10.3.7 Auto Positioning.....	14
10.3.8 Aligning Icons.....	15
10.3.9 Auto Snap Align	15
10.3.10 Sizing Objects.....	15
10.3.11 Swapping Connections	17
10.3.12 Transforming Icons, Labels, & Annotations	17
10.3.13 Changing Icons	18
10.3.14 PFD Navigation.....	19
10.3.15 Stream Routing	20
10.3.16 Rebuilding the PFD	25
10.3.17 Connecting Streams & Operations.....	25
10.3.18 Disconnecting Streams & Operations	29
10.3.19 Cut/Paste Functions	30
10.3.20 Stream Label Options.....	32
10.3.21 Annotations & Labels	34
10.3.22 Hiding PFD Objects	35
10.3.23 Printing the PFD	36
10.4 Graph Control	36
10.4.1 Data Page Tab.....	37
10.4.2 Axes Tab	38
10.4.3 Title Tab.....	39
10.4.4 Legend Tab.....	40
10.4.5 Plot Area Tab	40
10.5 Format Editor.....	41

10.1 Introduction

This chapter explains the commands that are available through the Edit menu and provides information on editing PFD objects.

You can access the Edit menu commands in three ways:

- Select the **Edit** menu in the menu bar.
- Use the **ALT** key in combination with the letter **E**.
- Use the **ALT** key by itself to move the active location to the **File** menu in the menu bar.

The up and down arrows move you 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.

10.2 Edit Menu

The functions listed under the Edit menu are available in all environments (i.e., Basis, Simulation, sub-flowsheet) and can be used across environments and outside of UniSim Design:



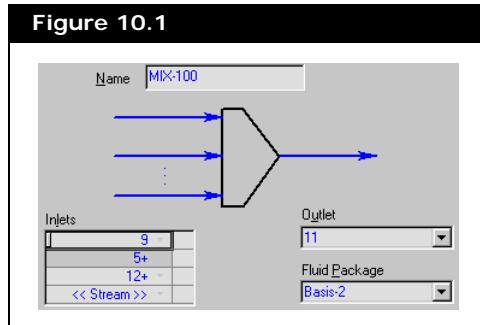
Edit menu

Command	Description
Cut	Copy and remove the selected values/objects from the current view.
Copy	Copies the selected values/objects to the clipboard.
Copy Special / with Labels	Copies the selected values/objects and their corresponding labels to the clipboard.
Paste	Place copied or cut selections in the location of your choice or in another application.

Although the edit functions are available throughout UniSim Design, not all areas within the environments support them. Only matrix type areas can be manipulated.

For example, you can copy a group of cells from the Workbook or from a table and paste them into the spreadsheet. In most areas, you can manipulate a group of cells by clicking and dragging them with the mouse. Whenever cells are grouped within a matrix (i.e., Inlet streams for a mixer as shown in the figure below), you can select more than a

single cell.



10.3 Editing the PFD

Any PFD in the simulation can be accessed from any location in the Simulation environment by clicking the PFD icon or using the **CTRL P** hot key.



PFD icon

The items being modified must be selected before using these tools.

The appearance of the PFD can be modified including all objects that are installed in the PFD:

- Streams
- Unit Operations
- Logical Operations
- Labels
- Text Annotations

In addition to selecting 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.

Use the following tools to modify the appearance of the PFD:

- Menu Bar
- Mouse and Keyboard
- Object Inspect menu
- PFD Inspection Menu

10.3.1 PFD Menu

The PFD menu is only available when the PFD is open and active. The



PFD menu

commands contained in the PFD menu are described in the table below:

Command	Description
Select Objects	Select multiple operations and streams on the PFD. The Select Objects view contains a filter for narrowing object selection. See Menu Bar Option for more information.
Show Hidden Objects	Hidden objects in the PFD can be viewed using this command. When selected, the Show Hidden Objects view appears. See Revealing Hidden Objects for more information.
Break a Connection	Breaks the connection between a stream and an operation without deleting either. See Disconnecting Using the Break Connection Tool for more information.
Swap Connections	Select two streams that are attached to the same operation and exchange their nozzle connections. See Section 10.3.11 - Swapping Connections for more information.
Auto Position All	Repositions all objects on the PFD to the best possible location as determined by the application. See Section 10.3.7 - Auto Positioning for more information.
Auto Position Selected	Auto positions only selected objects. See Section 10.3.7 - Auto Positioning for more information.
Select Mode	Select the operating mode for the PFD. This is either Move, Size, or Attach. See the PFD Modes section for more information.
Drag Zoom	Click and hold the mouse button to drag a frame around a region, then release the mouse button. The PFD is redrawn showing only that region.
Add a PFD	Adds a new page to the PFD Notebook. The command to clone an existing PFD is available.
Delete this PFD	Deletes the active PFD without a prompt to confirm the action. You cannot delete the PFD if it is the only one in the case.
Rename this PFD	Change the name of the PFD that appears on the tab.

10.3.2 Object Inspect Menu

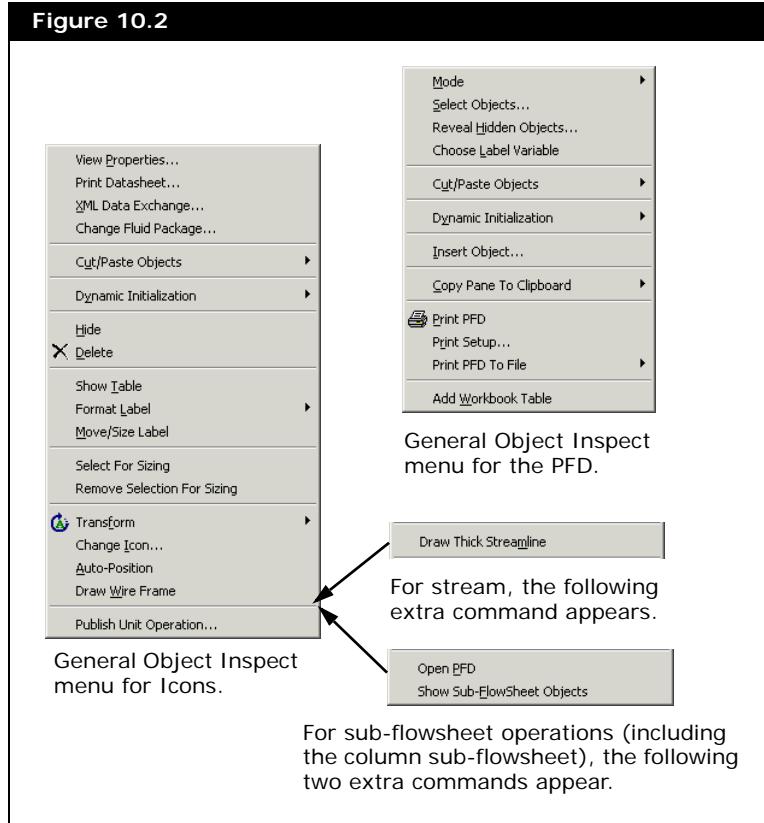
To access the Object Inspect menu of an icon, position the cursor over the icon and right-click.

Use the Object Inspect menu to manipulate the icons in the PFD. The menu options vary depending on the selected item (Stream, Operation, Column or Text Annotation).

The entire PFD Object Inspect menu can be accessed by right-clicking on an area of the PFD where there are no icons. The menu of available

commands appears.

A menu item with an arrow head pointing to the side has further commands on a sub-menu.



10.3.3 PFD Tools

Several tools are provided to help simplify interaction with the PFD. These tools are described in the following section.

Pan/Zoom Functions

For the PFD to respond to the keyboard command, it must be the active view.

The pan and zoom functions allow you to focus on a particular area within the PFD or to view all the objects within the flowsheet. The following table describes the available functions:

Function	Icon	Definition
Zoom Out		Zooms the display out by 25%. Click the Zoom Out icon, located in the lower left of the PFD, or use the keyboard command SHIFT PAGE DOWN .
Zoom In		Zooms the display in by 25%. Click the Zoom In icon, located in the lower left of the PFD, or use the keyboard command SHIFT PAGE UP .

Function	Icon	Definition
Zoom All		Displays all visible objects in the current PFD. Click the function icon, 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		Scroll the mouse wheel forward to display the PFD zoom in.
Zoom Out Mouse Wheel		Scroll the mouse wheel backward to display the PFD zoom out.
Zoom In HOME Key		Select PFD objects and press the HOME key to zoom in on those objects. If no objects are selected, the entire PFD is shown.
Toggle between last two Zoom views		Use the keyboard key Z .
Pan 15% Left, Right, Up, Down		Use the 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 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		Press the PERIOD key on the keyboard, and the PFD shifts, making the location of the cursor the centre of the view.

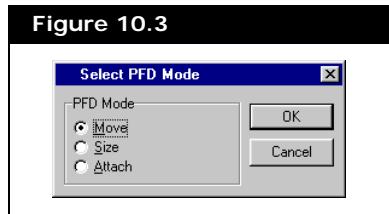
PFD Modes

The PFD operates in the following four modes:

- Move
- Attach
- Auto Attach
- Size

Only one mode can be active at a time, so when you are in attach mode, you cannot move or resize icons. There are three ways to change the PFD mode:

- Select the **Select Mode** command from the **PFD** menu to open the Select PFD Mode view. Use the radio buttons in the PFD Mode group to select the mode.



- Right-click the PFD, then select the mode options in the **Mode** sub-menu from the PFD Object Inspect menu.
- Use the icons in the PFD tool bar.

Name	Icon	Definition
Move/Attach Mode		Controls the Move and Attach modes in the PFD. Move is the default setting and is used to relocate selected operations and streams. When the icon is "pressed" you are in Attach mode and can connect streams and operations graphically. Refer to Section 10.3.17 - Connecting Streams & Operations for more information about Attach mode.
Auto Attach Mode		In this mode, new operations placed on the PFD automatically have their own required material and energy streams connected to them. These generated streams are automatically given a numerical value for a name. See the Auto Attach Mode section for more information.
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, drag the size of the box in any of these eight directions. Refer to the section, Section 10.3.4 - Selecting PFD Objects for more details on sizing PFD objects.

Auto Attach Mode

Auto attach mode does not attach energy streams already in the PFD to an added operation, even if the energy stream is selected and the operation requires an energy stream.

Auto attach mode always generates and attaches new energy streams for added operations that require energy streams.

In Auto Attach mode, any unit operation added to the PFD is automatically attached with the required material and energy streams. The generated streams are automatically named based on the user's defined naming preference (see [Section 12.2.4 - Naming Page](#) for more information on setting the naming preference).

The Auto Attach mode behaviour changes when a new unit operation is added, depending on what is selected in the PFD.

- When nothing is selected and a unit operation is added, then the required streams are created and attached.

- The Auto Attach functionality is based on an anchor stream(s). Once a stream(s) is selected, it is treated as the anchor. The behaviour from this point is dependent on the current connectivity of the anchor stream(s).

If the anchor stream is a product from a unit operation, it is automatically attached as the feed to the unit operation being added. The remaining streams for the unit operation are automatically created and connected.

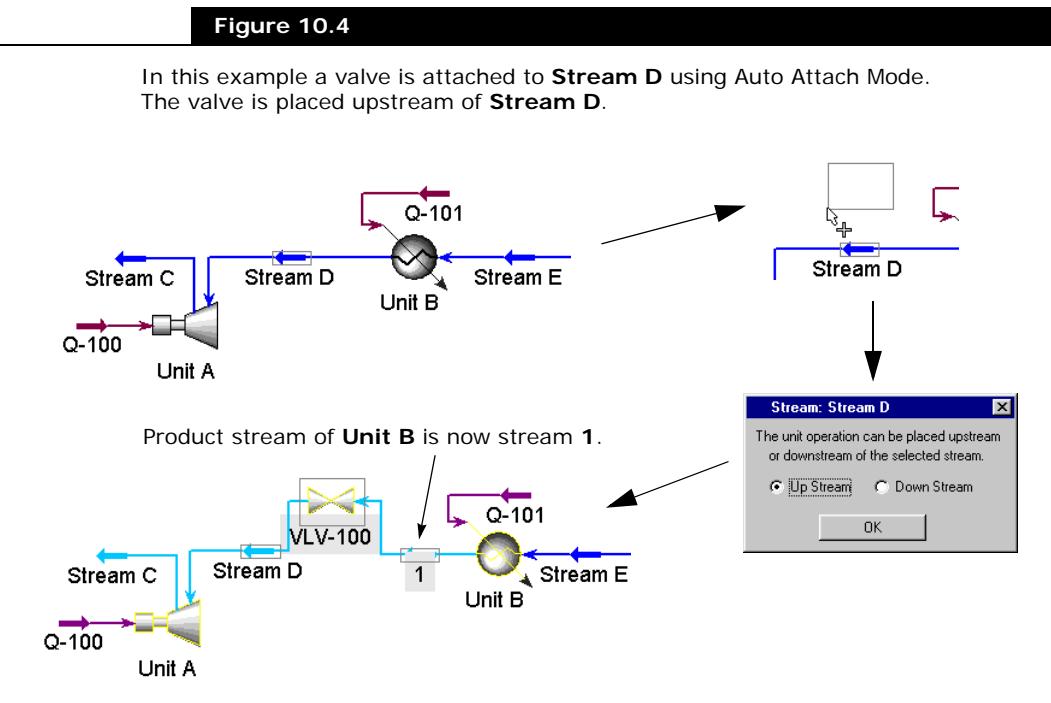
If the anchor stream is a feed stream to another unit operation, it is automatically attached as a product to the unit operation being added.

Since most unit operations have multiple product streams, the selected product stream is attached to the top product stream (i.e., usually the vapour stream).

If the anchor stream is both a feed and a product, you are prompted to insert the unit operation upstream or downstream of the anchor stream.

Figure 10.4

In this example a valve is attached to **Stream D** using Auto Attach Mode. The valve is placed upstream of **Stream D**.



- When multiple streams are selected, (in some cases) this mode attaches multiple feeds and products depending on what is selected (in other words, one feed and one product or three feeds and a product).

When one product and one feed are selected and a new unit operation is added, both the feed and product streams are

attached to the unit operation along with the remaining required streams.

If multiple feeds and a product are selected and a new mixer or separator is added, all the selected feeds and the product are attached to the unit op along with the remaining required streams.

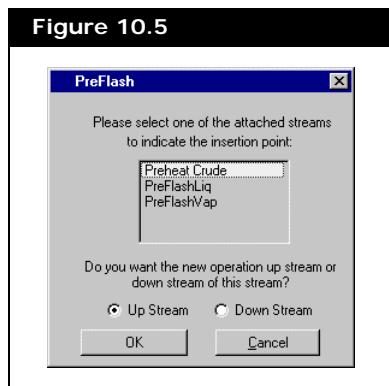
When multiple feeds and products are selected, multiple feeds are attached to the mixer or separator, but a new product stream is created because the system does not know which selected product stream attaches to where.

You can select multiple product streams and use Auto Attach mode to add a tee.

- Auto attach mode has limitations; with the sub-flowsheet and template, it creates all required streams and attaches them to the unit operations. Because there are so many different feeds and products, even if there is a feed or product selected, all new streams are attached, but not necessarily where you wanted them attached.
- You are not limited to selecting streams; you can also select a unit operation.

If you select a unit operation with a single feed and a single product and add a new unit op, then a view prompts you to add the unit operation upstream or downstream of the selected unit operation.

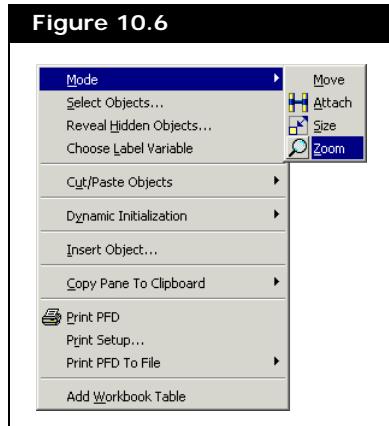
For example, the figure below displays a view prompting an object to be attached up or down stream:



If you select a unit operation with multiple feed or products and add a new unit operation, you are prompted to select a stream from a list of streams that are attached to this selected unit operation. You are also prompted to specify if the new unit operation is to be attached upstream or downstream of the stream selected from the attachment list.

Additional Icons

Name	Icon	Function
Break Connection		Break the connection between a stream and an operation. When you place the cursor over the stream you want to break, the cursor appears with a checkmark. Click any portion of the stream between the stream icon and the operation to break the connection. Refer to Section 10.3.18 - Disconnecting Streams & Operations for more details.
Swap Connections		Switches the nozzle connection points for two streams attached to the same operation. For more information, see Section 10.3.11 - Swapping Connections .
Drag Zoom		When you click this icon, the cursor becomes an arrow and magnifying glass combination. Click and drag around a region of interest to redraw the PFD showing only the selected region. You can also Zoom from the PFD Object Inspect menu by clicking Mode and then Zoom (see Figure 10.6) or by selecting Drag Zoom from the PFD menu in the menu bar.
Add Text Annotation		Adds text to a PFD. When clicked, a '+' symbol is added to the regular cursor and a rectangular box appears at the end of the pointer. Position the cursor where you want to place the text, click the mouse button, and then type the text into the view that appears. See Section 10.3.21 - Annotations & Labels , for more information.
Quick Route Mode		Move icons quickly about the PFD (in other words, object icons can be moved with their attached streams overlapping the other object icons). Turning off the Quick Route mode, enables UniSim Design to reposition the stream lines so that there is no overlap of the object icons. See Section 10.3.15 - Stream Routing for more information.
Drag Mode		Enables you to shift/scroll through the PFD view and see other areas of the PFD. To scroll across the PFD, click and drag the mouse cursor on the PFD. Similar functions as the scroll buttons, except with in the Drag mode you can scroll diagonally.
Object Palette		Enables you to access the Object Palette. See Section 8.1.1 - Install Objects Using the Object Palette for more information.
Colour Scheme		Displays the PFD Colour Schemes view. A new scheme can be created or an existing one selected, edited or deleted.



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

10.3.4 Selecting PFD Objects

When an object is selected, the icon is surrounded by a border and the label background is selected.

To manipulate the PFD, you must be able to select PFD objects such as streams, operations, and text annotations. You can select single or multiple objects, but you cannot be in Attach mode when doing so.

Press and hold SHIFT while clicking the objects to select scattered multiple objects in the PFD.

Single Object Selection

To select a single object in the PFD, do one of the following:

- Position the cursor over the object and click the mouse button. The object has a white rectangular box around its icon when selected.
- Use the keyboard by pressing **S** to cycle through all items in the PFD and **SHIFT S** to cycle backwards through all items.

Multiple Object Selection

There are three methods available to select multiple objects and they are as follows:

- Mouse drag option
- Menu bar option
- Keyboard/mouse option

Mouse Drag Option

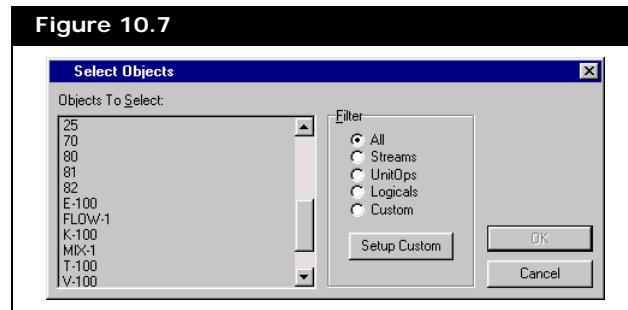
If selecting multiple objects within the same area, click and hold the mouse button while dragging a box around the entire group of objects. When all of the objects are within the box, release the mouse button. Each object is outlined to indicate that it is selected.

Menu Bar Option

1. From the **PFD** menu or the **PFD Inspection** menu, select the **Select Objects** command. The Select Objects view appears.
2. From the list of available objects, select the necessary objects. Select multiple objects by holding down the **CTRL** key and clicking each object being selected.

Figure 10.7

Use the filter for a more specific list of objects.



3. Click the **OK** button after all selections are made. This closes the view and returns you to the PFD. Each object is outlined to indicate that it is selected.

Keyboard/Mouse Option

1. Position the cursor (in the PFD) over the first object you want to select and click the mouse button.
2. To select a second object, hold down the **SHIFT** key and click the second object. The two objects are now selected.
3. Repeat step #2 until all the necessary objects are selected.

10.3.5 Deselecting PFD Objects

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

- Click any empty spot in the PFD.
- Press the **D** key to deselect all the selected items.
- Press the **SHIFT** key and click on a selected object to deselect only that one item.

10.3.6 Moving Objects

Objects can be moved in either Size or Move mode.

The internal Auto Snap feature can prevent an icon from being moved with the keyboard arrow keys. See [Section 10.3.8 - Aligning Icons](#) section for more information.

You can move objects individually or as a group.

1. Select the object or objects you want to move.
2. Position the cursor over one of the selected objects.
3. Press and hold the mouse button while dragging the cursor to the new position on the PFD. Release the mouse button. If multiple objects are selected, all selected objects move simultaneously.

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

10.3.7 Auto Positioning

Use the Auto Positioning function to automatically reposition streams and unit operations. Select the object(s) being repositioned and 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 object(s).

Auto positioning works differently depending on the object selected. For unit operations, a new location for the object is determined and it is placed 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 the stream is initially positioned by UniSim Design after the Auto Positioning function is accessed. If a unit operation icon that is attached to the stream is then moved, the stream is automatically repositioned, floating with the unit operation icon.

When you manually move a selected stream, the auto positioning function becomes inactive. Any subsequent movement of an operation that this stream is attached to does not affect the position of the stream. 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	From the PFD menu, select either Auto Position All or Auto Position Selected.
Object Inspect menu	Select Auto Position from the Object Inspect menu. Only selected items are repositioned.

10.3.8 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 is aligned.

1. Select the objects that you want to align. At least two objects must be selected.
2. Right-click the icon that is to be the anchoring point (all other selected objects are aligned with this icon).
3. Select **Align Selected Icons** from the Object Inspect menu.

10.3.9 Auto Snap Align

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

The Auto Snap feature is always active and cannot be toggled on and off. This poses a problem if you want to move an icon using the keyboard arrow keys. If UniSim Design detects that the object is within the set tolerance, the object initially moves in the direction of the arrow key, but snaps back to its original position. To overcome this, use the **SHIFT** key with the keyboard arrow key to move the object by larger increments.

10.3.10 Sizing Objects

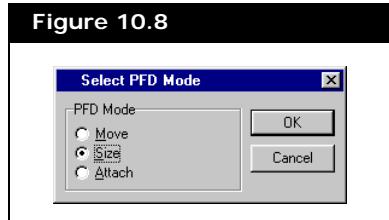
Operations and streams installed on a PFD are all a standard size. Sometimes the size of these objects needs to be changed; this can be done in Size mode only.

1. Activate the Size mode by doing one of the following:
 - Click the **Size Mode** icon in the PFD toolbar.



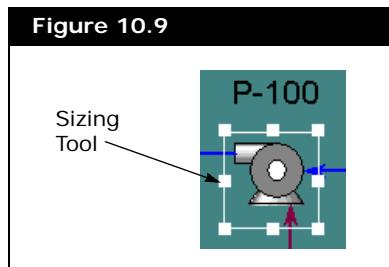
Size Mode icon

- Select the **Select Mode** command from the **PFD** menu. The **Select PFD Mode** view appears.



Click the **Size** radio button.

- Right-click in the PFD and select **Mode>Size** from the Object Inspect menu.
- Select the object being resized. A white outline with eight handles appears around the icon. These identify the directions in which you can size the icon.



- Place the cursor over one of the target handles. The pointer changes to the sizing tool (line with an arrowhead at each end) indicating the directions that the object can be stretched or compressed.
- Click and drag the mouse in the direction you want to size the object. The icon is sized according to the change in the size of the box.

You can size Labels and Annotations only 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 appears on one line.

The Pump shown in **Figure 10.9** is ready to size, however, the name associated with it, P-100, is not resized. It remains at its default size even though the size of its associated icon changes. To change the size of the label, change the Font size of the text. See the **Moving & Sizing Labels** section.

Rather than sizing several objects individually, you can size multiple objects simultaneously. Enter Size mode and select the objects you want to size. Each object has its own outline around it. Select any one of these boxes and then resize the object. All the selected objects are sized by this factor.

10.3.11 Swapping Connections



Swap Connections icon

-  Available to Swap Connections
 Unavailable to Swap Connections

This function lets you select two streams attached to the same object and swap their nozzle connections. This is useful when streams cross each other.

1. Activate the Swap Connections tool by doing one of the following:
 - Click the **Swap Connections** icon in the PFD tool bar.
 - Right-click a connection point and select **Swap Attachments** from the Object Inspect menu.
 - Select **Swap Connections** from the **PFD** menu.
 - Press the **F** hot key to activate the **Swap Connections** icon. The **ESC** key reverses the effect of the **F** key.
2. The cursor (when it is over an area of the PFD) takes on a special flip stream appearance, indicating which streams are available for the operation (arrow with a checkmark and a number 1).
3. Click the first stream you want to swap and the cursor changes (replacing the 1 with a 2).
4. Click the second stream being swapped.
5. After the swapping is complete, the **Swap Connections** icon is released.

UniSim Design indicates an unacceptable choice for Swap Connections by replacing the checkmark with an X.

10.3.12 Transforming Icons, Labels, & Annotations

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

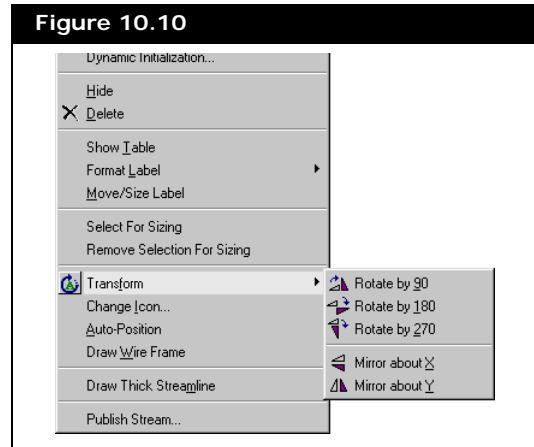
When you add a unit operation, stream, or annotations to the PFD, the icon appears as shown in the Object Palette. You may want to alter the orientation of an icon to improve PFD clarity. This can be done using rotating and mirroring functions.

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.

You can use the Rotate and Mirror functions to change the orientation of multiple objects at the same time.

Access the Rotate and Mirror functions from the Object Inspect menu.

1. Select the object(s) you want to transform.
2. Right-click on one of the objects to open the Object Inspect menu.
3. Select **Transform** to open a sub-menu containing the Rotate and Mirror functions. Click the required function.

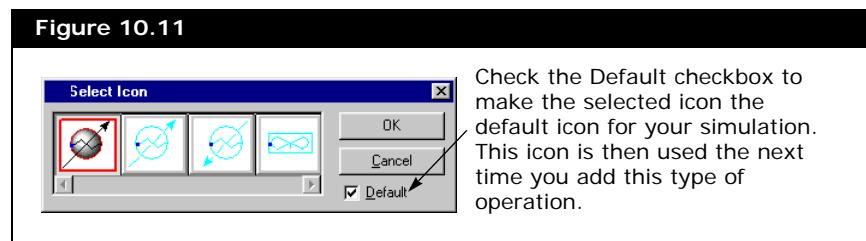


10.3.13 Changing Icons



UniSim Design always displays the default icons for the PFD. However, there are optional icons that can be used to represent the same unit operation.

1. Select the unit operation for which you want to change the icon.
2. Right-click on the unit operation to open the Object Inspect menu.
3. Click **Change Icon** (if there are no alternate icons, this command is disabled in the menu). This opens the Select Icon view.
4. Click the icon you want to use. The figure below shows the options available for the Cooler operation.

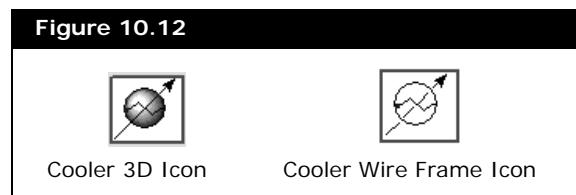


5. Click **OK** to accept the selection and return to the PFD.

If you have more than one icon selected in the PFD and they represent the same type of operation, then you can change all selections to an alternate icon at the same time. If the operations are of different types, the Change Icon command is not available in the Object Inspect menu.

Wire Frame & 3D Icons

When you right-click a three dimensional icon, the Draw Wire Frame command is available in the Object Inspect menu. Any operation or stream with a three dimensional icon can be transformed to an outline representation, or a Wire Frame. A wire frame of the Cooler is shown in the figure below.



Alternatively, the Object Inspect menu of a wire frame object contains the Draw 3D command, which draws the icon in its default 3D view.

To switch all objects into wire frames, do the following:

1. Select all objects.
2. Right-click one of the objects and select the Draw Wire Frame command in the Object Inspect menu.

Thick Stream Line

The Draw Thick Streamline command (in the stream Object Inspect menu) creates a more visible stream by making it wider than the other streams. This command is useful when tracking one particular stream in a complex flowsheet (i.e., a pipe network).

Return a stream to its default thickness by right-clicking the stream and clicking Draw Normal Streamline from the Object Inspect menu.

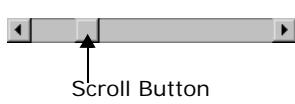
10.3.14 PFD Navigation

Auto Scrolling

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

Auto scrolling enables continuous horizontal or vertical scrolling of the active pane, depending on the location of the mouse.

To initiate auto scrolling, do the following:



1. Place the cursor anywhere in the active view.
2. Press and hold the mouse button and drag the mouse 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 begins to scroll. The boundary closest to the cursor determines the direction of scrolling.

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

Mouse Wheel Scrolling



The Static PFD Scroll cursor



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). Click the mouse wheel while the cursor is in the PFD. The cursor changes to the static PFD scroll cursor. Point the mouse in the direction you want to scroll. After you have reached the location that you want 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).

10.3.15 Stream Routing

Quick Route Mode

The Quick Route function can be accessed in any mode.

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

When you use the Quick Route function, UniSim Design relocates and connects the objects without considering the other objects in the flowsheet. For example, if moving a valve, its icon and streams are relocated without repositioning the streams, even if one passes over another icon.

After exiting Quick Route mode, the streams are repositioned automatically so they do not overlap the icons. This means the streams are repositioned once instead of after relocating each object.



Quick Route icon

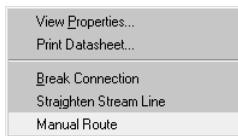
To access the Quick Route function, click the Quick Route icon in the PFD tool bar. After manipulating the objects on the PFD, click the Quick Route icon again to ensure that no streams overlap unit operation

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



To exit the Manual Route function, click an empty area of the PFD.

Full Manual Route mode can only be accessed by right-clicking a stream (not the stream arrow icon). From the Object Inspect menu, select the Manual Route command.

Only the portion of the stream that is inspected becomes available for manual routing (i.e., right-click either the portion upstream or downstream of the stream icon).

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 the 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. The bend points are not shown in this case.

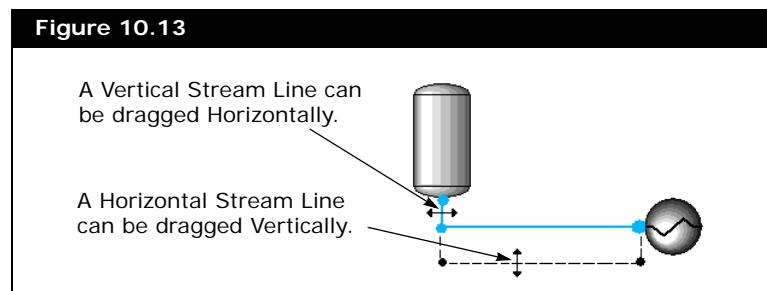
Moving Line Segments

To move a portion of a stream, do the following:

1. Right-click the stream portion and select **Manual Route** from the Object Inspect menu. Anchor points appear at each corner of the stream route.

When you move the pointer over the stream, the pointer changes to a double arrowhead cursor.

This cursor is oriented in the direction in which you can move the selected stream portion; **Vertical** for a horizontal section and **Horizontal** for a vertical section.



2. Click the mouse button and drag the stream portion to the new location. As you move, a thin black line appears, indicating the new stream route.
3. Release the mouse button when you reach the target location. The stream is redrawn through the new path.

Adding Bend Points

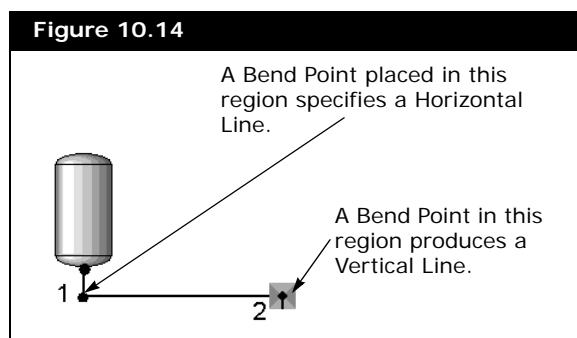
A bend point is an anchor that the stream passes through.

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

Place a bend point by clicking the stream.

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

The idea of bend points is illustrated in the figure below.



Bend point 2 is the one being manipulated. You can initially create a horizontal or vertical line segment and subsequent line orientation is determined by that first line, i.e., horizontal segments follow vertical and vice versa.

1. Select an existing bend point to begin the new route. The cursor changes to an arrow with a '+' symbol at the end when placed over a bend point.
2. Around a bend point there are four regions, two that 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, a light coloured line shows the area where the new line routing is placed.
3. Click to place a new bend point.
4. Continue to move the cursor to the location of the next bend point and place it by clicking the PFD.

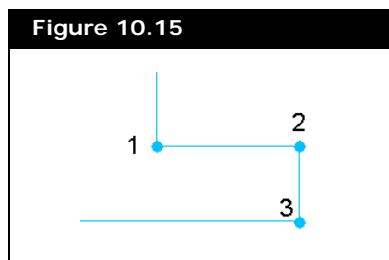
The mouse cursor displays a checkmark at an acceptable location for the final Bend Point.

For slight kinks in a stream it could 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.

4. Manual Routing can only be completed by placing the final bend point on an existing bend point, otherwise the new routing you just laid out does not appear in the PFD. Any bend points added are erased.

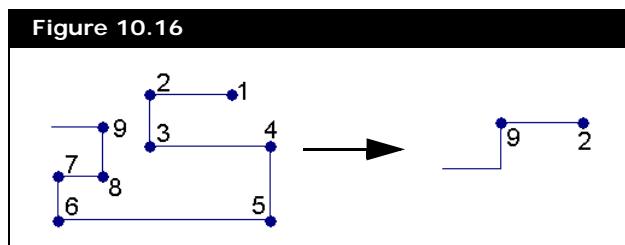
Removing Bend Points

In certain instances, bend points can be removed to provide a more direct route between operations. When there is an extra bend in the route, as shown in the figure below, close the section.



Select either the horizontal section between points 1 and 2 and move it vertically until points 1 and 3 coincide, or select the vertical section (2 to 3) and move it horizontally until points 1 and 2 coincide. Either route results in the extra bend being removed from the stream.

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 the figure below, point 2). Next, select the bend point at the end of the section (point 9) and double-click on the end point. All intermediate bend points are removed.



Connection Line Straightening

The Connection Line Straightening function removes all bend points from a stream to straighten the line between the stream icon and the unit operation. To access this function, the orientation of the Stream icon must align with the nozzle connection (in other words, a horizontal stream icon is in alignment with a nozzle connected to the side of a unit

The end of the thin red line that is not selected serves as the anchor point, and does not move when the line is straightened.

operation).

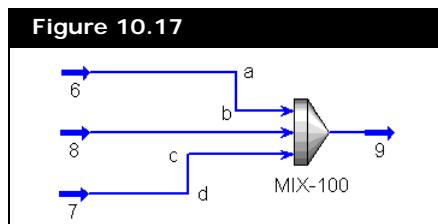
To straighten a connection line, do the following:

1. Right-click the stream and select **Straighten Stream Line** from the Object Inspect menu.
2. The stream section being straightened becomes a thin red line and the cursor changes to the acceptable/unacceptable connection indicator.
3. Move the cursor to either end of the red line where the cursor changes from an X to a checkmark, indicating an acceptable choice for line straightening.
4. Click to straighten the line.

Line Segment Alignment

While performing manual routing, the mouse can be used to align sections of streams. Horizontal sections are aligned horizontally and vertical sections are aligned vertically. The streams do not need to be connected to the same unit operation, but the stream sections must be in close 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 is used. The Mixer, shown in the figure below, has three feed streams, two that contain vertical sections. For presentation purposes the vertical sections of the streams are to be aligned.



To align segments ab and cd, do the following:

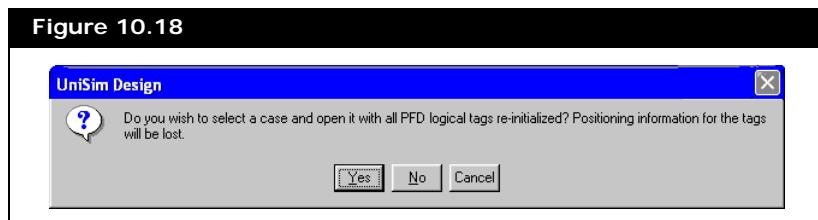
1. Right-click the line segment **ab** and select the **Manual Route** command. The entire line becomes selected and its bend points are shown.
2. Click the anchoring segment, in this case segment **cd**.

The segments are now aligned. Follow the same steps for the alignment of horizontal segments.

10.3.16 Rebuilding the PFD

In addition to manipulating the PFD, you can also rebuild the PFD for any simulation case.

1. Before opening the case you want to rebuild, use the hot key combination **CTRL SHIFT K Z** to display a message asking if you want to rebuild the PFD.



2. Click the **Yes** button. The Open Simulation Case view appears.
3. Open the case for the PFD you want to rebuild. If there is more than one PFD (in the main and sub-flowsheet environments), you are asked if you want to rebuild each PFD (in the main and sub-flowsheet environments).
4. Click **Yes** if you want to rebuild the specified PFD and **No** if you do not. If there is only one PFD (i.e., no sub-flowsheets), you are only asked once if you want to rebuild the main PFD.

10.3.17 Connecting Streams & Operations

There are two ways to connect streams to an operation:

- In the operation property view, select the streams' name in the inlet and outlet field/cell. New streams can be generated and connected to the operation by entering the new stream's name in the inlet and/or outlet field/cell.
- On the PFD view, make the connections in Attach mode.

Connecting Streams & Operations in Attach Mode

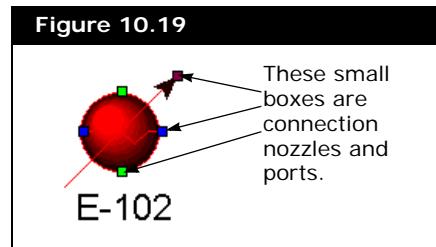
The Attach mode can be used to connect an operation to an existing stream, generate a new stream connected to an existing operation, or

create a new stream to connect two operations.

Connection nozzle or port colours:

- Dark Red - Energy Stream
- Blue - Material Stream
- Green - 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.



When the PFD is in Attach mode, connection nozzles and ports automatically appear on each icon as the cursor passes over the icon. In addition to the coloured boxes that indicate the different connection types, a Fly-by description appears when the cursor is placed over a connection nozzle or port.

Creating a New Stream from an Operation

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



Stream Connection Tool cursor

To create a stream from an operation in Attach mode, do the following:

1. Place the cursor over the required operation connection point or nozzle. When the cursor is in the correct location, a white square appears at the tip of the cursor.
2. Click and drag the mouse to an empty space in the PFD. Keep dragging the stream until the Stream Connection Tool cursor appears at the end of the stream. If you release the mouse button while a full black square is visible, the stream will not be created.
3. Release the mouse button and a stream icon appears. UniSim Design names the stream using the next stream name as defined under the current Session Preferences.

Refer to [Chapter 3 - Streams in the UniSim Design Operations Guide](#) for more information about streams.

Connecting Operations to Existing Streams

In Attach mode you can connect streams operations or vice versa on the PFD. The procedure for both is identical.

To connect an operation to an existing stream, do the following:

1. Place the cursor over the required connection point, then click and drag the cursor to the target stream. A line indicating the creation of

See [Section 10.3.15 - Stream Routing](#) for information on manually changing the route of a stream.

a stream appears as you move the mouse.

Both Streams and Operations have nozzle(s) and port(s). The connection points that activate depend on the origin of the connection (i.e., when connecting from a stream outlet, only nozzles to operations appear).



Stream Connection Tool cursor

As you approach the stream, the available connection activates. When you are within the defined connection region for a nozzle (larger than the region used when UniSim Design indicates available connection points), the cursor changes to the Stream Connection Tool cursor.

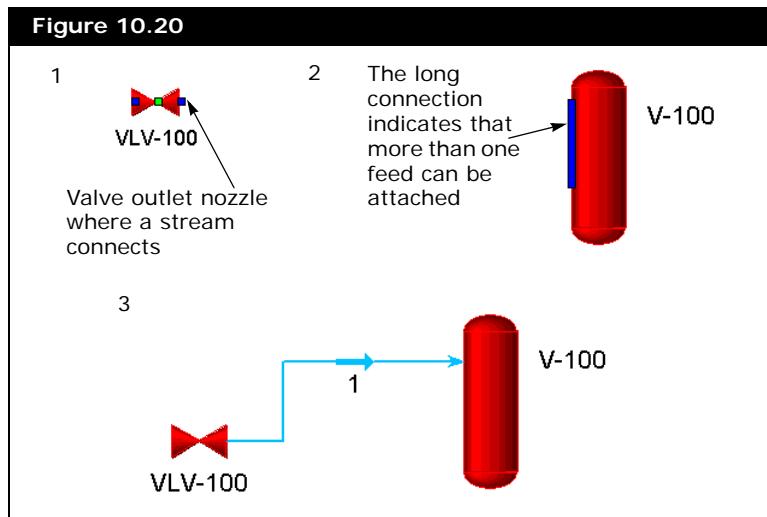
2. To complete the connection, release the mouse button when a white square appears at the tip of the cursor.

Connecting Two Operations

Directly connecting two operations automatically creates a new stream using the next available name as defined in the Session Preferences. To accomplish this, you must make the connection on the PFD in Attach mode.

1. Select the connection point of the operation to which the stream will be connected (in the case of [Figure 10.20](#) the Valve, VLV-100).
2. Click and drag the new stream toward the operation that the stream is connected to (the separator V-100 in [Figure 10.20](#)). The available connection points are indicated. In this case, because the stream is taken from the outlet of the Valve, only the connection point on the inlet area of the Separator appears.

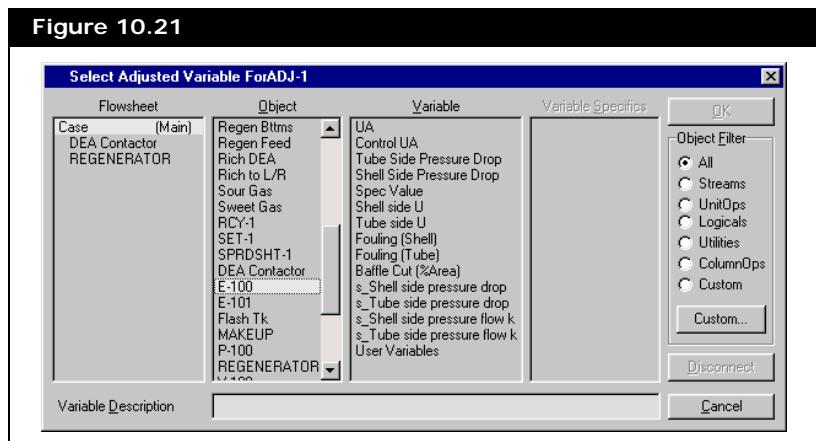
3. When you reach the connection region, the cursor changes to a solid white square. Release the mouse button to complete the connection and create a new stream.



Connecting Logical Operations

Refer to [Chapter 12 - Logical Operations](#) in the [UniSim Design Operations Guide](#) for more information about this view.

Logical operations can be connected to operations or streams in the same way as other objects. The only difference is that after the connection is made, the Select Adjusted Variable view appears. This view varies depending on the type of logical operation and whether the connection is made to a stream or operation.



10.3.18 Disconnecting Streams & Operations

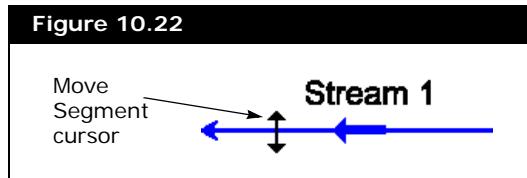
Connections between streams and operations can be broken by doing one of the following:

- Enter the operation's property view and break the connection by deleting the stream's name from the inlet or outlet cell/field.
- Break the connection on the PFD by using the Object Inspect menu or the Break Connection tool.

Disconnecting Using the Object Inspect Menu

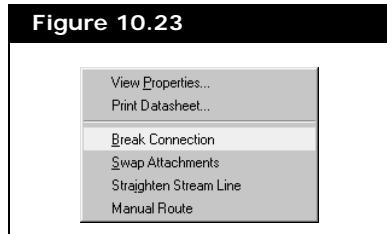
Use this procedure to break a connection using the Object Inspect menu.

1. On the PFD, move the cursor to the stream where you want to break the connection (anywhere except over the stream icon).
2. When the cursor is over the stream line, it changes into the Move Segment cursor. Right-click the mouse button.



The Swap Attachments command appears in the Object Inspect menu only when more than one stream is attached to a certain location (in other words, 2 feed streams).

3. From the menu that appears, select **Break Connection**. (Depending on the selected stream, the Object Inspect menu may not have all of the commands as shown in the figure below.)



Disconnecting Using the Break Connection Tool

Breaking the connection does not delete the stream, only its connection to the operation.

The Break Connection tool lets you break an existing stream connection in the PFD. You can only break one connection at a time. If you want to break a second connection, select the Break Connection option again. You can either break an inlet or an outlet stream connection, depending on which side of the stream icon you select.



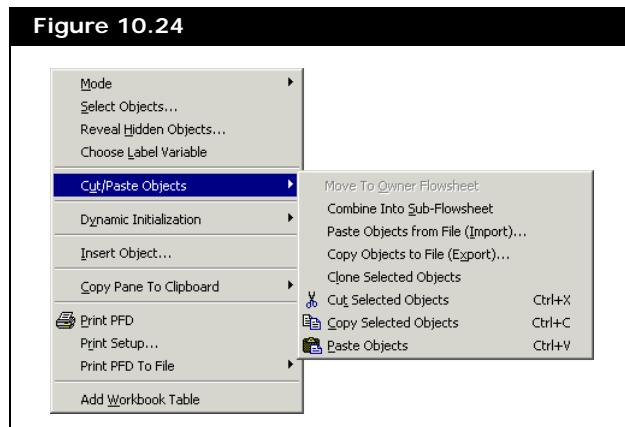
Break Connection icon

Use this procedure to break a connection using the Break Connection tool:

1. Access the Break Connection tool by doing one of the following:
 - Select **Break a Connection** from the **PFD** menu.
 - Click the **Break Connection** icon in PFD tool bar.
2. Move the cursor to the stream where you want to break the connection. When the cursor is in position, it changes from an **X** to a checkmark.
3. When the checkmark appears, click the stream. The connection is broken.

10.3.19 Cut/Paste Functions

The Cut/Paste Objects sub-menu available in the PFD Object Inspect menu provides several commands for adding, removing, and recombining flowsheet objects.



Creating Sub-Flowsheets

The Combine Into Sub-flowsheet command lets you select multiple PFD objects in your simulation case (in the main flowsheet) and create a

sub-flowsheet containing those objects.

Use this feature to organize complicated flowsheets. For example, you can divide your flowsheet into different sections to make the information more readable.

1. Select the PFD objects (i.e., unit operations, streams and logical operators) to be included in the new sub-flowsheet.
2. Right-click to open the Object Inspect menu.
3. From the **Cut/Paste Objects** sub-menu, select the **Combine Into Sub-flowsheet** command.

Moving Sub-Flowsheet Objects Back to the Parent Flowsheet

1. Right-click the sub-flowsheet icon. The Object Inspect menu appears.
2. From the Cut/Paste Objects sub-menu, select the **Move Contents to Owner Flowsheet** command.

Even though the contents of the sub-flowsheet were moved to the owner flowsheet, the sub-flowsheet still exists, but can be deleted. If you decided to “re-collapse” the sub-flowsheet objects, a new sub-flowsheet is created.

Importing/Exporting Objects

You can export flowsheet objects from one PFD and import to another using the Import Objects and Export Objects commands in the PFD Object Inspect menu. The objects that you export or import are saved as an UFL file. See [Section 4.3 - UFL Files](#) for more information.

Exporting Objects

Exported object files use the extension.ufl.

1. Select the PFD objects you want to export.
2. Right-click on one of the selected objects.
3. From the Cut/Paste Objects sub-menu, select Export Objects. The File Save view appears.
4. Enter a name and destination for the flowsheet file.
5. Click **Save**.

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

Importing Objects

Templates can be imported into a flowsheet by using the Import Objects command in the PFD Object Inspect menu. All basis information already supplied to the flowsheet is automatically applied to the imported objects. None of the basis information from the case in which the objects were exported is saved in the template file.

1. Right-click the PFD to open the Object Inspect menu.
2. From the Cut/Paste Objects sub-menu select Import Objects. This opens the Open File view.
3. Browse to the location of the flowsheet file (*.ufl) you want to open and select it.
4. Click **Open**.

Cloning Objects

You can clone flowsheet objects on your PFD using the Clone Selected Objects command in the Copy/Paste Objects sub-menu. All object information is automatically cloned into a new set of objects. Only the object name changes.

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. You cut or copy an object(s) in one flowsheet and paste the object(s) to another location on the PFD or into any sub-flowsheets of any case. If the destination sub-flowsheet was created using a different fluid package, then some of the copied information may not be transferrable and will be omitted.

10.3.20 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 appears in place of each stream name.

Common Variable Choices

There are some hot key combinations that let you toggle between

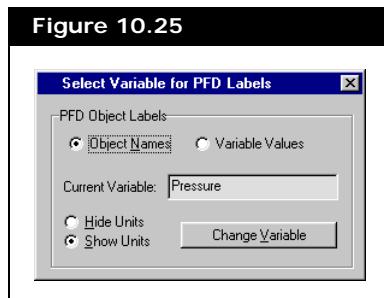
stream name labels and some of the more common stream variables:

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

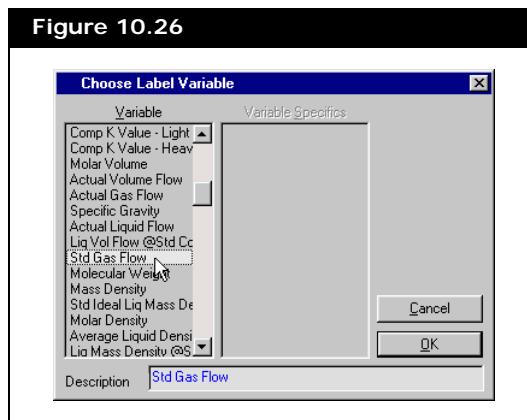
Other Variables

You can also replace the stream name labels with other variable values. To change the stream name label to the Std Gas Flow, do the following:

1. Right-click in a blank part of the PFD to open the Object Inspect menu.
2. Select **Choose Label Variable** to open the Select Variable for PFD Labels view.



3. Click the **Change Variable** button to open the Variable Navigator.
4. Select **Std Gas Flow** from the list of available variables.



5. Change the variable description in the Description field, if required.
6. Click **OK**.

Use the **Hide Units** and **Show Units** radio buttons to toggle the units on and off.

You are returned to the Select Variable for PFD Labels view. The variable you just selected appears in the Current Variable field.

7. Click the **Variable Values** radio button to display the values of the current variable on the PFD. Select the **Object Names** radio button to display the names of the streams on the PFD.
8. Click the **Close** icon.

10.3.21 Annotations & Labels

Annotations are any text that appears on the PFD aside from the object labels. UniSim Design allows you to enter annotations anywhere on the PFD. UniSim Design automatically labels an object on the PFD with the object's name. The object's name is taken from the name that appears on the Name field in the object's property view.

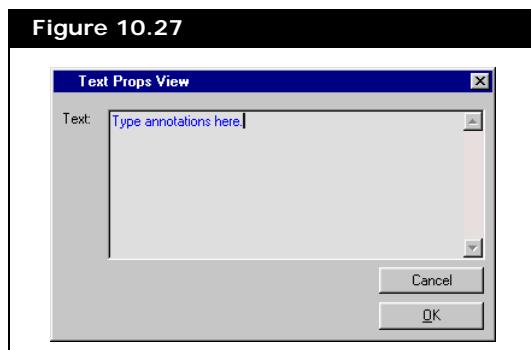
Adding Annotations

Use this procedure to add text to the PFD.



Add Text Annotation icon

1. Click the **Add Text Annotation** icon in the PFD tool bar.
2. Move the cursor to the location on the PFD where you want to place the text and click the mouse button. The Text Props view appears.
3. In the **Text** field, type the text that you want to appear on the PFD.
4. Click the **OK** button.



Editing Annotations

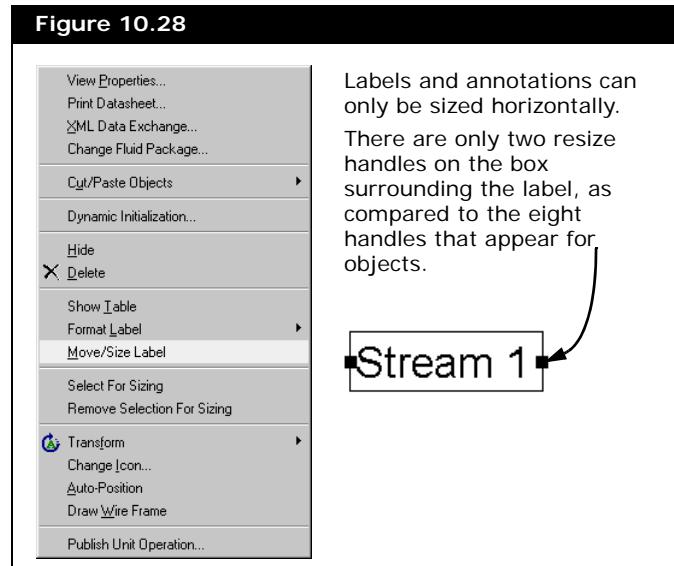
1. In the PFD, right-click the Text Annotation, then click the View Properties command. The Text Props view appears.
2. In the **Text** field, modify the text, then click **OK**.

Other editing options include the following:

- **Hide.** Hides the selected Label or Text Annotation.
- **Delete.** Deletes the selected Label or Text Annotation.
- **Transform.** Rotate by 90, 180 or 270 degrees.
- **Change Font.** Changes the font for a Label or Text Annotation. This function is not global; it changes only the selected object's font.
- **Change Colour.** Opens the colour palette and lets you change the colour of the selected Label or Text Annotation.

Moving & Sizing Labels

You can move and size object labels on the PFD. Right-click any icon and select the Move/Size Label command. You can also 'free' the Label by selecting the object and then pressing **L** on the keyboard. Move labels the same way you move operations, streams, and annotations.



Once you select the Move/Size Label command, the label is unlocked from the object to which it belongs. The label re-locks itself once it is deselected, however, you can select the label's corresponding object icon and then select the label again without re-locking the Label. This facilitates moving and sizing both the icon and its label at one time without repeatedly selecting the Move/Size Label command.

10.3.22 Hiding PFD Objects

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

Any object on the PFD can be hidden, and you can hide multiple objects at one time. Hiding an object does not prevent the case from solving. You can hide operations, streams, and text annotations by selecting the Hide command from the Object Inspect menu.

A Show/Hide option also exists for displaying sub-flowsheet objects on the main flowsheet PFD. For details, see [Section 7.24.4 - Access Column or Sub-Flowsheet PFDs](#).

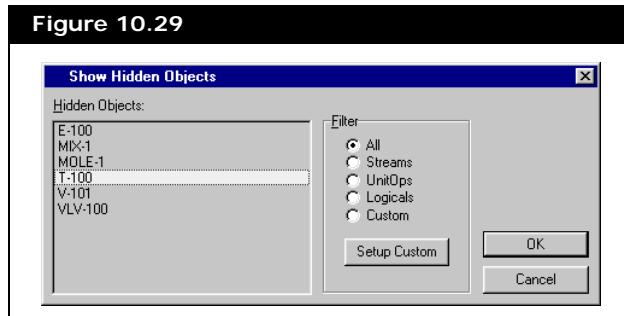
To hide labels, you need to select the Format Label command and then select Hide Label from the sub-menu.

Revealing Hidden Objects

Use this function to reveal any objects that are hidden on the PFD. You can access this function in two places:

- PFD menu
- Object Inspect menu

When you select the Show Hidden Objects command, the Show Hidden Object view appears. This view allows you to specify the hidden objects to be revealed. Select the objects you want to show, then click the OK button.



10.3.23 Printing the PFD

For more information see [Section 9.2.3 - Printing the PFD](#).

All objects (Streams, Operations, Text, and PFD Tables) included within the PFD view can be printed. UniSim Design prints the PFD as it appears on the screen.

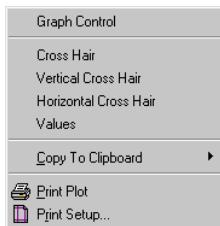
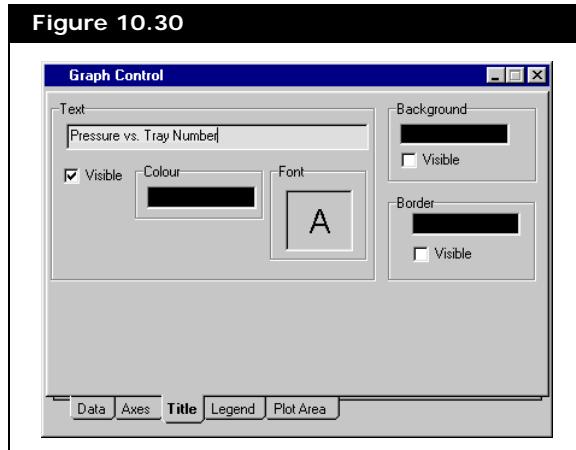
10.4 Graph Control

The changes made to a plot using the Graph Control tool are specific to the active plot.

To make global changes that affect all plots, use the Colours and Fonts pages on the Resources tab of the Session Preferences property view.

You can customize each individual plot in UniSim Design using the Graph Control tool. You can also modify many of the plot characteristics, which are grouped into the five tabs of the Graph

Control property view:



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

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

While the plot area has focus, you can also click and drag on either the Legend or Title to reposition the selected item.

10.4.1 Data Page Tab

For each data set on the plot, you can do the following:

- View the data set Type
- Modify the data set Name
- Specify the Colour and Symbol that represent the data on the plot
- Select a Line Style
- Show/hide the Symbol or Line
- Show/hide the name in the Legend

Any changes that are made affect only the data set that is selected in the list.

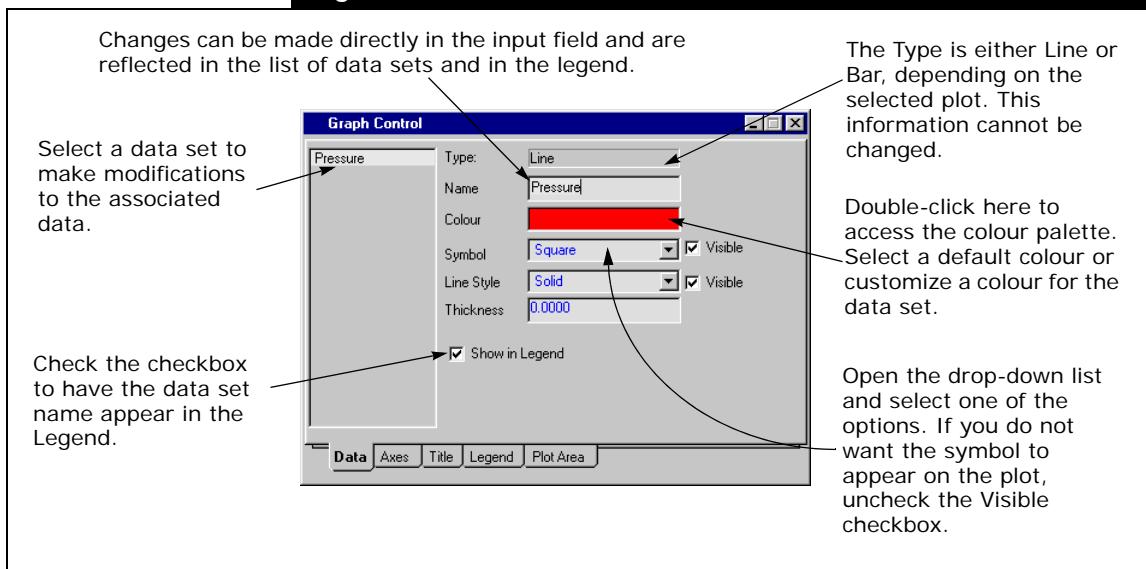
To modify the appearance of a data set, select the name of the set in the list of data sets. The information that corresponds to the selected

All changes instantaneously affect the plot. There is no need to close the Graph Control property view to see the modifications.

The Symbol drop-down list is not available for Bar Charts.

data set appears, as shown in the figure below.

Figure 10.31



10.4.2 Axes Tab

Refer to [Section 10.5 - Format Editor](#) for information regarding value formatting.

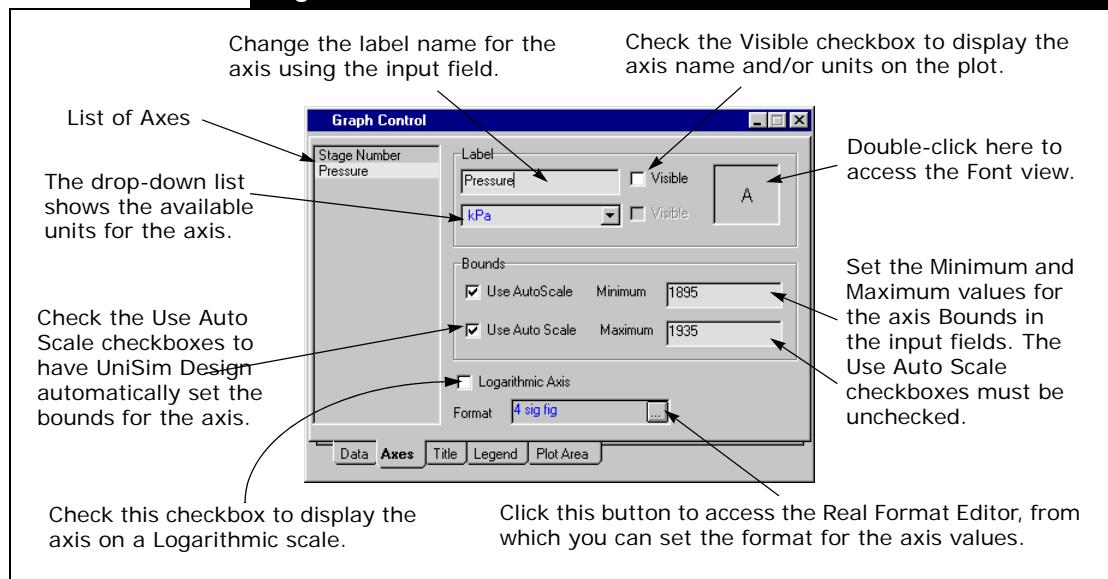
From the Axes tab, you can do the following for each axis:

- 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.

Any changes that are made affect only the axis that is selected in the list. To make modifications to the appearance of an axis, select the name in the list of axes. The information that corresponds to the

selected axis appears.

Figure 10.32

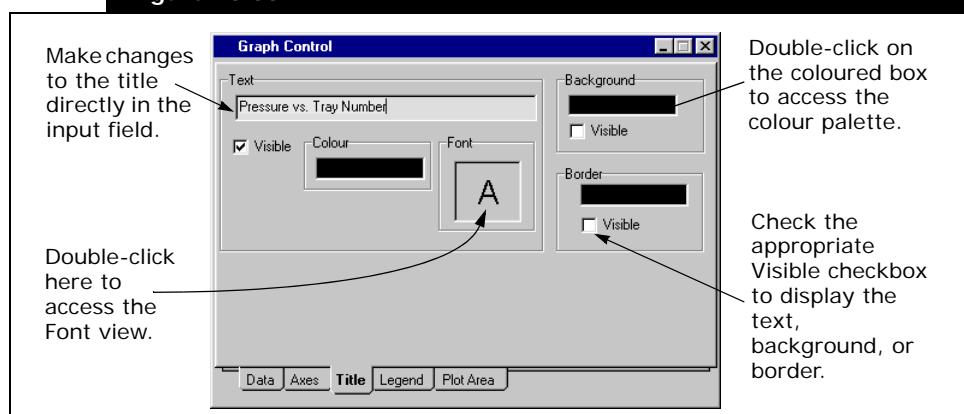


10.4.3 Title Tab

On the Title tab, you can do the following:

- 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.

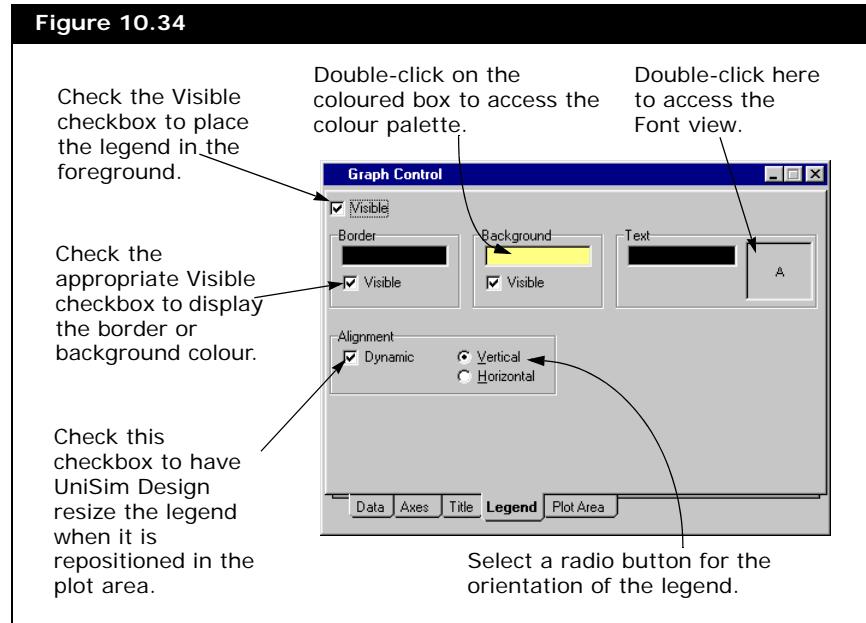
Figure 10.33



10.4.4 Legend Tab

The Legend tab allows you to change the appearance and location of the legend. You can do the following:

- Change the colour of the border, background, or text.
- Select 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.

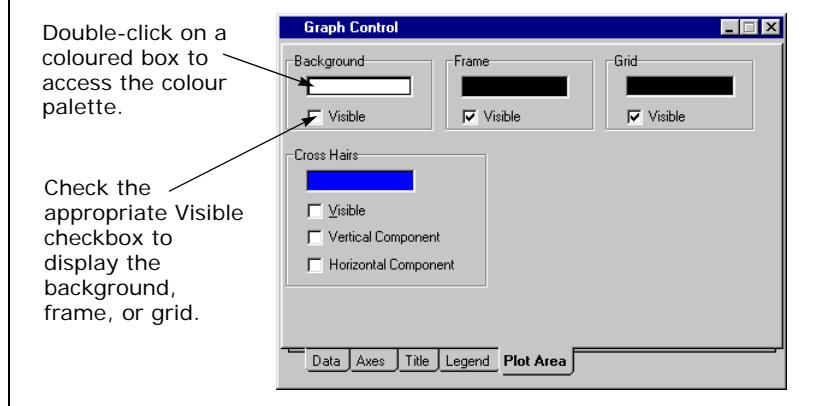


10.4.5 Plot Area Tab

From the Plot Area tab, you can do the following:

- Change the colour of the background, frame, and grid.

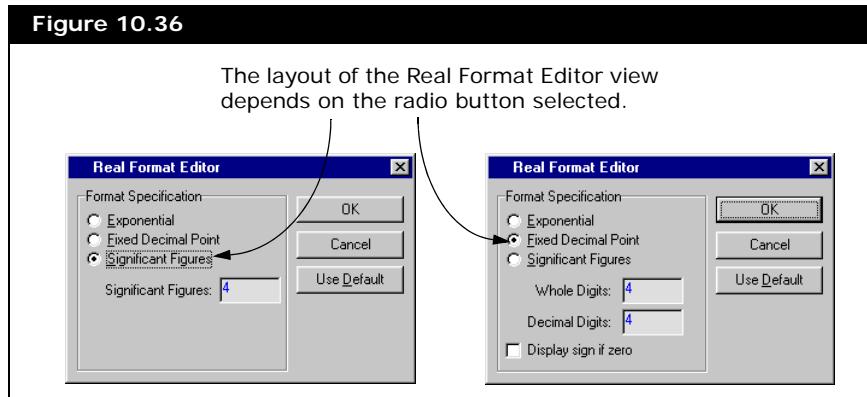
- Show/Hide the background, frame, and grid.

Figure 10.35

10.5 Format Editor

The Real Format Editor can be accessed in the following locations:

- Workbook Setup property view
- Graph Control property view

Figure 10.36

From the view in the figure above, you can set the format of values displayed in UniSim Design. The Format Specification group uses three

radio buttons for the options available:

Format	Description
Exponential	The values are in exponential form with a specified number of Significant digits. For example, 8546 appears as 8.546e+03 if 4 was specified in the Significant cell.
Fixed Decimal Point	Specify the maximum number of digits that appear before the decimal point in the Whole cell (see above figure). If the Whole cell limit is exceeded by a value in UniSim Design, exponential form is used. In the Decimal Digits cell, input the number of digits that appear after the decimal point. If you check the Display sign if zero checkbox, UniSim Design displays a '+' symbol in front of a value that appears as zero using the current precision. For example, a composition of 0.000008 appears as zero when using a Decimal Digits value of 4. With the checkbox checked, the cell shows +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 want to display.

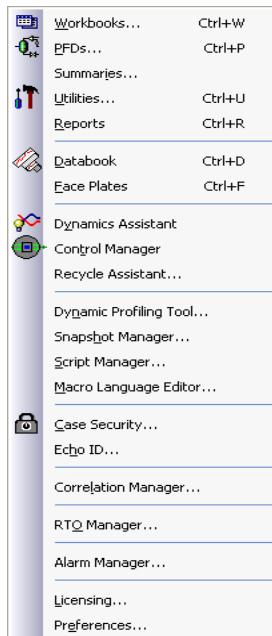
The Use Default button closes the Real Format Editor view and assigns the UniSim Design default format to the associated values.

11 Simulation Tools

11.1 Introduction	3
11.2 Workbook	3
11.3 PFD.....	3
11.4 Case Summary.....	3
11.5 Utilities	4
11.6 Reports	4
11.7 Databook	4
11.7.1 Databook Variables.....	5
11.7.2 Process Data Tables.....	7
11.7.3 Strip Charts.....	13
11.7.4 Data Recorder.....	27
11.7.5 Case Studies	30
11.7.6 Spec Scenarios	37
11.8 Face Plates	39
11.9 Dynamics Assistant.....	40
11.10 Control Manager	41
11.11 Dynamic Profiling Tool.....	41
11.11.1 Profiling a Case	42
11.12 Snapshot Manager	44
11.12.1 Local Snapshots Tab	44
11.12.2 Real Time Monitor.....	46
11.12.3 External Snapshots	47
11.13 Script Manager	50
11.13.1 Recording a New Script	51
11.13.2 Script Playback	52
11.14 Macro Language Editor	52
11.15 Case Security	54
11.15.1 Locking a UniSim Design Case.....	54
11.15.2 Loading a Locked UniSim Design Case.....	56

11.15.3 File Security Setup	56
11.15.4 Unlocking a Case.....	64
11.15.5 Runtime Mode.....	65
11.16 Echo ID.....	70
11.17 Correlation Manager	71
11.17.1 Adding Property Correlations to Streams	73
11.17.2 Removing Property Correlations from Streams.....	74
11.17.3 Cloning Property Correlations.....	74
11.17.4 Deleting Cloned Property Correlations	75
11.17.5 Adding Correlation Sets to Streams.....	75
11.17.6 Deleting a Correlation Set.....	77
11.17.7 Gas Properties Correlation	77
11.17.8 RVP Properties	82
11.18 Alarm Manager	85
11.18.2 Understanding the Alarm Manager	86
11.19 Variable Navigator.....	88
11.19.1 Using the Variable Navigator	89
11.20 Simulation Balance Tool.....	90
11.21 RTO Manager	90
11.21.1 Adding a Transfer Table	91
11.21.2 Viewing a Transfer Table	92
11.21.3 Deleting a Transfer Table	93
11.21.4 Adding Object	93
11.21.5 Removing Object	94

11.1 Introduction



Tools menu

This chapter provides information about the tools available in the Tools menu. To access the **Tools** menu commands, use one of the following methods:

- Click the **Tools** menu in the menu bar.
- Press **ALT T**.
- Press just the **ALT** key to move the active location to the **File** menu in the menu bar.

When you press the **ALT** key, the menu bar is active and you can navigate 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.

11.2 Workbook

The Workbook command opens the UniSim Design Workbook. Refer to [Section 7.23 - Workbook](#) for more information on using the Workbook.

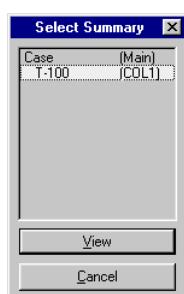
11.3 PFD

The PFD command opens the UniSim Design PFD. This section is covered in [Chapter 8 - UniSim Design Objects](#) and [Chapter 7 - Simulation Environment](#).

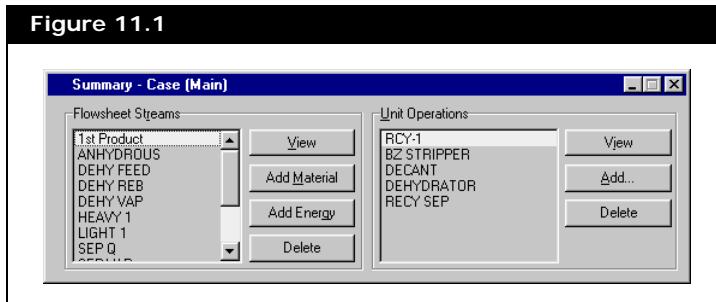
11.4 Case Summary

To open the Select Summary view, do the following:

1. Select the **Summaries** command from the Tools menu.
2. From the list of available flowsheets, select the flowsheet you want to view.



- Click the **View** button to display the Summary view. The Summary view provides an alternative way of adding, editing, and deleting streams and unit operation in specific flowsheets.



Select more than one object at a time by holding down the **CTRL** key and then clicking each object you want to select.

The Flowsheet Streams group is used for adding, editing, and deleting streams in the selected flowsheet. The stream list contains all of the streams available in the selected flowsheet.

The Unit Operations group is used for adding, editing, and deleting unit operations in the selected flowsheet. The unit operation list contains all of the unit operations available in the selected flowsheet.

Upon deleting a stream or unit operation, you are prompted to confirm the deletion.

11.5 Utilities

The Utilities command opens the Available Utilities view. Refer to [Section 7.26 - Utilities](#) for more information on utilities.

11.6 Reports

The Reports command opens the Report Manager. Refer to [Section 9.3 - Reports](#) for more information on using the Report Manager.

11.7 Databook

There is only one Databook in each UniSim Design case, which contains variables from all flowsheets.

The Databook is used for systematically analyzing data and lets you monitor key process variables in both Steady State and Dynamics modes. Variables for all Databook features are selected in a single location and can be activated from the main list for each application.

To access the Databook, use one of the following methods:

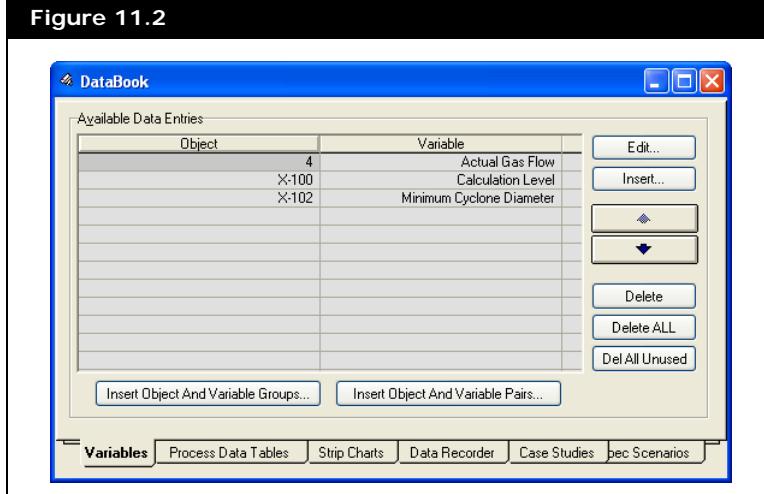
- Select the **Databook** command from the **Tools** menu.

- Press the **CTRL D** hot key combination.

11.7.1 Databook Variables

Attach as many variables to the Databook as required.

All variables used by the Databook are managed through the Variables tab.

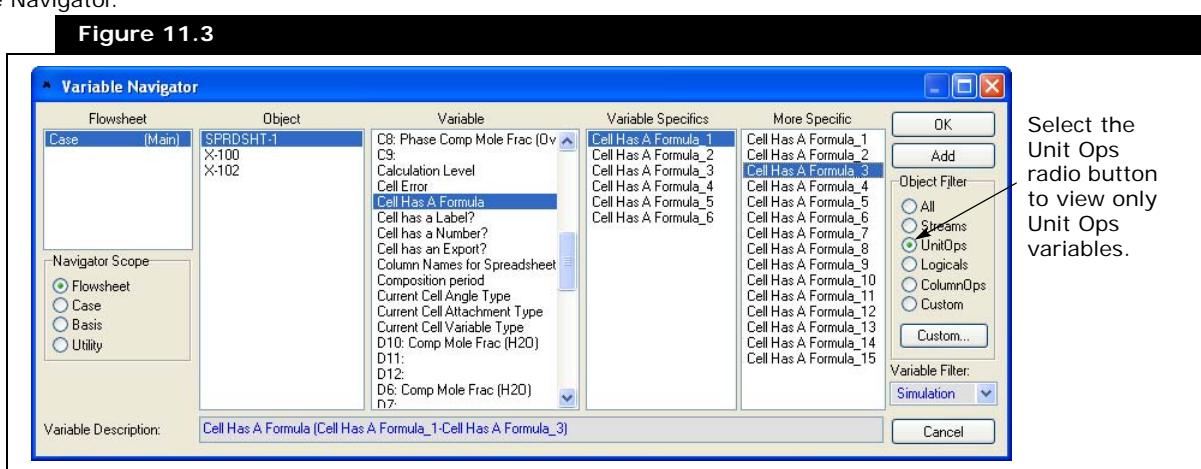


Adding a Variable

Refer to [Section 11.19 - Variable Navigator](#) for information regarding the Variable Navigator.

Use this procedure to add a variable to the Databook.

- Click the **Insert** button to display the Variable Navigator view.



- From the list of available flowsheets, select the flowsheet in which the object is located.

Click the **Cancel** button at any time to close the Variable Navigator

without accepting any changes.

3. From the list of available objects, select the object containing the variable.
4. From the list of available variables, select the variable you want to add. Some variables (e.g., Comp Mass Flow) require that you select a variable specific.
5. Click in the Variable Description field and type a description for the variable (optional).
6. Click **OK**. The variable appears in the list of available data entries on the **Variable** tab.

Another way of adding variables is via the Insert Object and Variable Groups or Insert Object and Variable Pairs buttons.

Editing a Variable

Refer to [Section 11.19 - Variable Navigator](#) for information about the Variable Navigator.

Use this procedure to edit a variable in the Databook.

1. From the list of available data entries, select the variable you want to edit.
2. Click the **Edit** button. The Variable Navigator view appears. This view lets you change the flowsheet, object, variable and description of the variable.
3. Make the required changes, then click **OK**, or click the **Cancel** button to close the view without making changes.

Deleting a Variable

Use this procedure to delete a variable from the Databook.

1. From the list of available data entries, select the variable you want to delete.
2. Click the **Delete** button

You will not be prompted to confirm the deletion, so ensure you have selected the correct variable.

When a variable is deleted, it is removed from all features in the Databook.

You can also Delete all Unused variables from the databook. this will remove variables which are no longer used by any of the Data Tables, Strip charts, Data Records, Case Studies or Spec Scenarios.

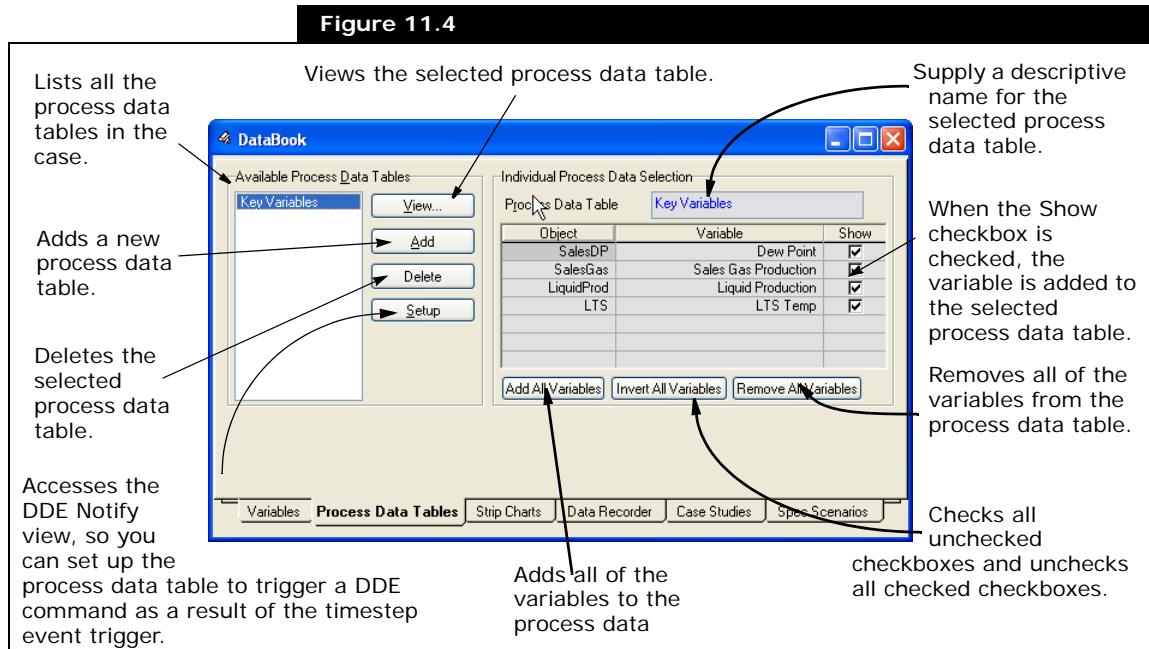
Optionally, you can also Delete All Variables. this will remove the variables from all the Data Tables, etc that are using the variables.

11.7.2 Process Data Tables

Variables for the process data table are selected on the Variables tab of the Databook.

Process data tables are installed individually through the Process Data Tables tab. Use this tab to view, add, or delete customized process data tables. For each table, add any combination of key process variables from the list of available variables. Variables can be used in multiple process data tables.

Figure 11.4



Adding a Process Data Table

1. Click the **Add** button. A process data table with default name **ProcData1** appears in the list of available tables. If required, type a new name in the Process Data Table field.
2. Check the **Show** checkbox for each variable you want displayed in this process data table.

Note that for UniSim Design models that are connected to external models, either UniSim Operate or PDTGen (refer to UniSim Operations Engineering Guide) can automatically generate Process Data Tables.

Viewing a Process Data Table

1. From the list of available process data tables, click the process data

Click the View Databook button to open the Databook if it is closed.

table you want to view.

2. Click the **View** button. The Process Data Table view appears.

Note that for UniSim Design models that are connected to external models, either UniSim Operate or PDTGen (refer to UniSim Operations Engineering Guide) can automatically generate Process Data Tables.

Figure 11.5



The screenshot shows a Windows application window titled "ProcData2 Data". The window contains a table with the following data:

Object	Variable	Value	Units	Tag	Access Mode
4	Actual Gas Flow	<empty>	ACT_m3/h	No Tag	No Transfer
X-100	Calculation Level	500.0		No Tag	No Transfer
X-102	Minimum Cyclone Diameter	0.3000	m	No Tag	No Transfer

At the bottom of the table, there are sorting options: "Sort by: Object Tag".

The Process Data Table shows the following for each variable:

- Name of the Object that the variable is attached to
- Object variable name
- Current value
- Units of the value
- Tag name of connected variable
- Access mode
 - No Transfer
 - Read
 - Write
 - Read/Write.

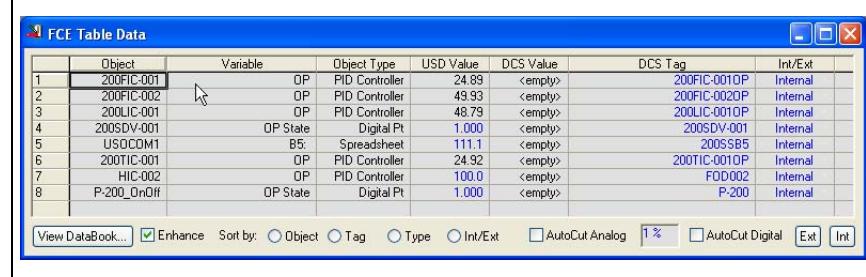
A radio button at the bottom of the table gives the user the option to sort the variables by object name or tag name.

Viewing the FCE Process Data Table

If the UniSim Design case is connected to an external model, it may be necessary to synchronize values from the external model to values in the UniSim Design model. This is commonly called integration. For example, if the UniSim Design model is used in an operator training simulator, there may be an external controls model. The outputs of the external controllers will have to be integrated with final control elements (FCE) in the UniSim Design model. The values from the external model and connection points in the UniSim Design model will be displayed in the Process Data Table named "FCE Table".

To facilitate the integration process, you can use the enhanced PDT.

Figure 11.6



The screenshot shows a software interface titled "FCE Table Data". It displays a table with columns: Object, Variable, Object Type, USD Value, DCS Value, DCS Tag, and Int/Ext. The table contains 8 rows of data. Row 1 is highlighted with a blue border. The "Object" column lists various objects like 200FIC-001, 200FIC-002, etc. The "Variable" column lists corresponding variable names. The "Object Type" column includes PID Controller, OP, Spreadsheet, and Digital Pt. The "USD Value" and "DCS Value" columns show numerical values. The "DCS Tag" column lists tags such as 200FIC-001OP, 200FIC-002OP, etc. The "Int/Ext" column indicates internal or external mode. At the bottom of the window, there are several buttons: "View DataBase...", "Enhance" (checked), "Sort by", "Object", "Tag", "Type", "Int/Ext", "AutoCut Analog", "AutoCut Digital", and "Ext" and "Int" buttons.

Object	Variable	Object Type	USD Value	DCS Value	DCS Tag	Int/Ext
1 200FIC-001		OP	PID Controller	24.89	<empty>	200FIC-001OP Internal
2 200FIC-002		OP	PID Controller	49.93	<empty>	200FIC-002OP Internal
3 200LIC-001		OP	PID Controller	48.79	<empty>	200LIC-001OP Internal
4 200SDV-001		OP State	Digital Pt	1.000	<empty>	200SDV-001 Internal
5 USOCOM1	B5	Spreadsheet		111.1	<empty>	200SB5 Internal
6 200TIC-001		OP	PID Controller	24.92	<empty>	200TIC-001OP Internal
7 HIC-002		OP	PID Controller	100.0	<empty>	FOD002 Internal
8 P-200_OnOff		OP State	Digital Pt	1.000	<empty>	P-200 Internal

The enhanced FCE Table shows the following for each variable:

- Name of the object that the variable is attached to.
- Object variable name.
- Object type (PID controller, Digital Pt, etc.)
- Current value in USD.
- Current value in the external model
- Tag name of connected variable (can be generated automatically by PDTGen)
- Internal / external mode of the USD variable. If the object type is a PID controller, digital point, selector, or transfer function, this will be the internal/external mode of that controller. If the object is a spreadsheet (or other non-controller), then this will be a variable internal to the Process Data Table to allow that object to function as if it had an internal/external mode.

The following options are shown at the bottom of the table:

- Switch between the normal and enhanced view of the table
- Sort by:
 - Object name
 - Tag name
 - Object type
 - Internal / external mode of the USD variable
- AutoCut Analog = automatically switch from internal to external mode for any variable in the table that is within a certain tolerance.
- The tolerance for the AutoCut option
- AutoCut Digital = automatically switch from internal to external mode for any digital point in the table where the USD value matches the external value.

A typical work process to integrate an external DCS system to a UniSim Design model would be as follows:

1. Run the integrator
2. Call up the FCE table
3. Sort by Internal / external mode so that variables in internal mode are on top

4. Set the tolerance to a small value (1% is the default)
5. Select AutoCut Analog
6. For the variable at the top of the list, look at the DCS tag (column 6) and call up that tag on the external DCS
7. Put the DCS controller in manual mode
8. If the DCS controller is not PV tracking:
 - Double click on the USD object name (column 1) to call up the view for that object
 - Set the DCS setpoint equal to the PV of the USD controller
9. Set the output of the DCS controller to equal the output of the USD controller (in column 4).
10. The USD controller will automatically switch from internal to external mode.
11. Put the DCS controller in its appropriate mode (automatic or cascade for example).
12. Make sure that the controller does not go unstable. You could check stability by making a small change to the setpoint on the DCS controller.
13. Move on to the next object in the FCE table.

The entire model can be integrated from the FCE table.

Viewing the MV Process Data Table

If the UniSim Design case is connected to an external model, it may be necessary to synchronize values from the external model to values in the UniSim Design model and vice versa. This is commonly called integration. For example, if the UniSim Design model is used in an operator training simulator, there may be external controls model. The measured variables (PV) in the UniSim Design model will have to be integrated with the external controllers measured variables. The values from the UniSim Design model and connection points in the UniSim Design model will be displayed in the Process Data Table named "MV Table".

To facilitate the integration process, you can use the enhanced MV

Table.

Figure 11.7



The screenshot shows a Windows application window titled "MV Table Data". The main area is a grid table with the following data:

	Object	Variable	Value	Units	Tag	Access Mode	Use Malf	Malf Status
1	FIC-100	Malfunction: Amplitude	0.0000	kg/h	No Tag	No Transfer	<input type="checkbox"/>	Normal
2	FIC-100	Malfunction: Delay Time	0.0000	seconds	No Tag	No Transfer	<input type="checkbox"/>	Normal
3	FIC-100	Malfunction: Fail Offset	300.0	kg/h	No Tag	No Transfer	<input checked="" type="checkbox"/>	Failed
4	FIC-100	Malfunction: Fail Status Of	0.0000		No Tag	No Transfer	<input type="checkbox"/>	Normal
5	FIC-100	Malfunction: Fail Status Va	0.0000	kg/h	No Tag	No Transfer	<input type="checkbox"/>	Normal

At the bottom of the window, there are several buttons and checkboxes:

- Enhance (checkbox checked)
- Sort by: (radio buttons for Object, Tag, Use Malfun, Malfunction Status)

The enhanced MV Table shows the following for each variable:

- Name of the object that the variable is attached to.
- Object variable name.
- Indicated value in USD.
- Tag name of connected variable (can be generated automatically by PDTGen).
- Use Malfunction checkbox to specify whether the variable is malfunctionable or not.
- Overall Malfunction Status of the variable.

The following options are shown at the bottom of the table:

- Switch between the normal and enhanced view of the table
- Sort by:
 - Object name
 - Tag name
 - Use Malfunction
 - Malfunction Status

If Use Malfunction checkbox is checked the data entry is malfunctionable for the entries other than PID Controller's PV. If the data entry is PID Controller's PV the entry is malfunctionable if the "Use Malfunction" checkbox is checked on the Malfunction page. If the data entry is PID Controller's PV then Malfunction Status is the Overall Malfunction Status of the data entry. Double click on the Use Malfunction cell to bring up the Malfunction view. The Malfunction view for the data entries other than PID Controller's PV is shown in **Figure 11.8**. If the data entry is the "PV" of PID Controller, the Malfunction view is same as the Malfunction page view appearing on the PID

Controller as shown in **Figure 11.9**.

Figure 11.8 Malfunction View of data entries other than PID Controller's PV

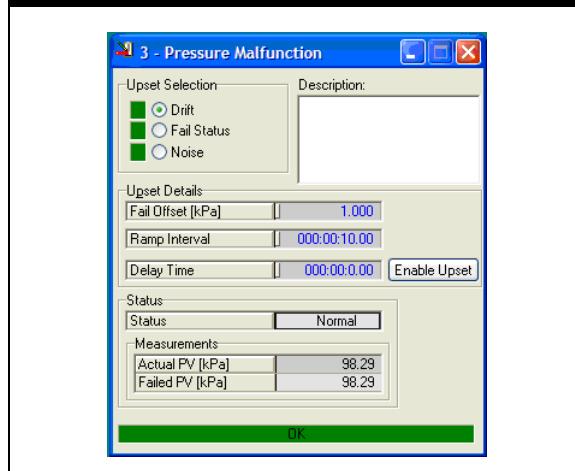
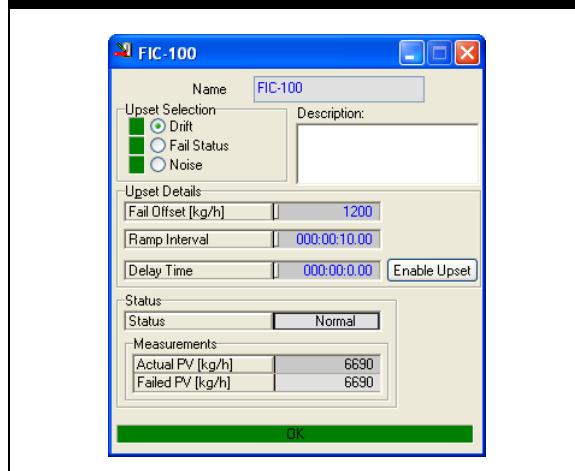


Figure 11.9 Malfunction View of PID Controller's PV



The malfunction details can be seen in the Malfunction view. The Malfunction view Status Bar is updated with Overall Malfunction Status when malfunction is enabled.

When Use Malfunction checkbox on the MV Table is toggled the malfunction details are not persisted for the data entries other than PID Controller's PV.

The variable value on the MV Table is the indicated value in USD. This value is the failed value of the variable. If the malfunction is not enabled, the failed value is same as the actual value and when the malfunction is enabled the failed value is calculated. The failed value calculations are detailed in the PID Controller's Malfunction section.

The PV value of the PID Controller is the failed value and not the actual value. The value of the variable within the object is the actual value for the data entries on the MV Table other than PID Controller's PV. The failed values are shown on the "Value" column in the MV Table.

Hence the value in the "Value" column (failed value or indicated value) should be used as the MV (measured value) and not the variable value within the object for the data entries other than PID Controller's PV.

Note: The malfunction of the MV simulates the malfunction of transmitter (transmitter may or may not physically be present in the flowsheet) which transmits the MV from the source module to the controller. The failed value of the MV is used by the controllers to determine the SP. The failed value of the MV is not used to update the property in the unit operation window. For example, if a stream's temperature is failed using the MV table, the stream's temperature is not updated with the failed value, but the failed value in MV table is transmitted as the PV to the controller thereby simulating the transmitter malfunction.

Deleting a Process Data Table

1. From the list of available process data tables, click the process data table you want to delete.
2. Click the **Delete** button.

You will not be prompted to confirm the deletion, so ensure you have selected the correct variable.

11.7.3 Strip Charts

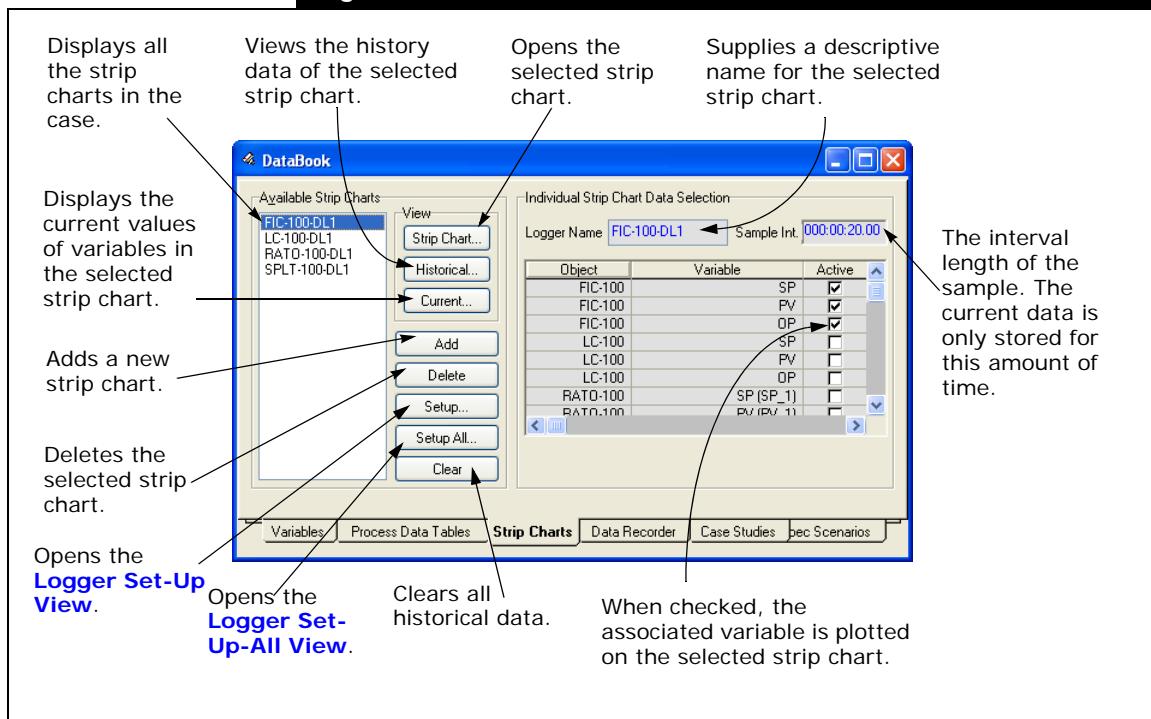
Variables for the strip chart are selected on the Variables tab of the Databook.

Use the strip chart tool to monitor the response of key process variables during dynamics calculations. Strip charts let you monitor the behaviour of process variables in a graphical format while calculations proceed. Current and historical values for each strip chart are also tabulated for further examination.

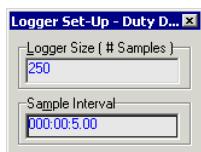
Strip charts are installed individually using the Strip Charts tab. You can have multiple strip charts, with each having an unlimited number of variables charted. The same variable can be used in more than one Strip Chart, so the use of multiple strip charts with a maximum of six

variables per strip chart is recommended.

Figure 11.10



Logger Set-Up View



The Logger Set-Up view allows you to modify the following options of the selected strip chart in the Available Strip Charts group:

- The amount of data that the strip chart keeps and displays in the graph.
- The frequency the strip chart records and displays the data.

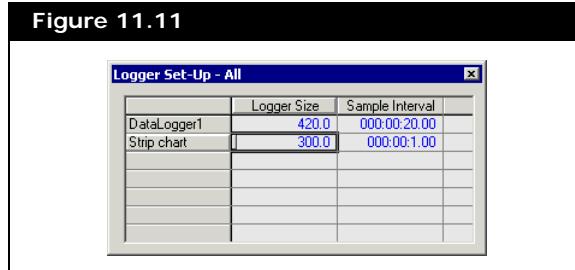
For example, if the Logger Size has a value of 250 and the Sample Interval has a value of 5 seconds, the strip chart stores and displays 250 values, and each value was recorded every 5 seconds. So the amount of time range displayed in the strip chart is 1250 seconds or 20 minutes and 50 seconds.

Logger Set-Up-All View

The Logger Set-Up-All view allows you to modify the following options of all the strip charts in the Available Strip Charts group:

- The amount of data that the strip chart keeps and displays in the graph.

- The frequency the strip chart records and displays the data.



Clear

Historical data can dramatically increase the size of a simulation case when it is stored to disk (and held in computer memory). You can clear all of the historical data by clicking the Clear button.

Adding a Strip Chart

Use this procedure to add a strip chart to the Databook.

1. Click the **Add** button to display a strip chart with default name DataLogger1 in the list of available strip charts. If necessary, type a new name in the Logger Name field.
2. Click the **Active** checkbox for each variable that you want to display in this strip chart.

Variables can exist in more than one strip chart.

Deleting a Strip Chart

Use this procedure too delete a strip chart from the Databook.

1. From the list of available strip charts, click on the strip chart you want to delete.
2. Click the **Delete** button.

You will not be prompted to confirm the deletion, so ensure you have selected the correct strip chart.

Viewing a Strip Chart

Use this procedure to view a strip chart.

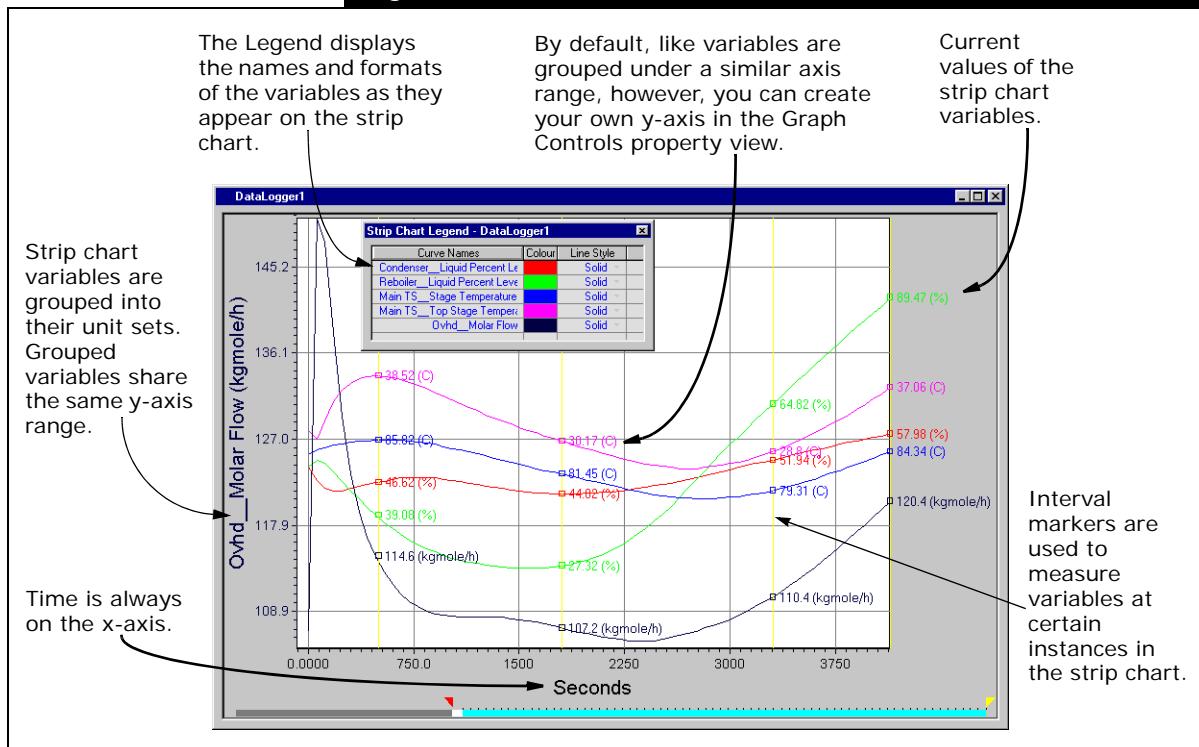
1. From the list of available strip charts, select the strip chart you want

When the Integrator is active, the strip chart begins to accumulate data points.

to view.

- Click the **Strip Chart** button. The Strip Chart view. appears.

Figure 11.12



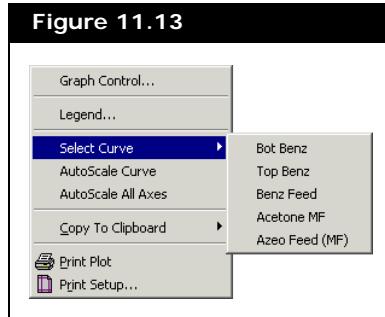
You can manipulate the information displayed on the strip chart within the Strip Chart view. There are several functions you can perform directly on the strip chart; these are described in the following sections.

Selecting Curves

The y-axis displays the range and units of a specific strip chart variable depending on the variable selected. Select a curve using one of the following methods:

- Click any part of the variable curve in the strip chart.

- Right-click the strip chart and click the **Select Curve** command, then from the sub-menu select the required variable.



Manipulating the Y-Axis Range

Use this procedure to manipulate the y-axis range of the strip chart.

- Position the cursor over an empty space in the strip chart.

Cursor	Description
	When the cursor is in the background of the strip chart, the following cursor appears. You can move along the strip chart in any direction.
	If you simply want to move vertically across the strip chart, move the cursor on or near a curve and the following cursor appears.

- Click and hold the mouse button until a multi-directional cursor appears.
- Drag the strip chart up if you want to display a lower range of values on the y-axis, or, drag the strip chart down if you want to display a higher range of values.

By default, strip chart curves are grouped into their unit sets. For example, 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 want to associate a different range with a variable in the strip chart, you must first create your own axis. Refer to the [Graph Control](#) section for more information.

Manipulating the X-Axis Range

Scroll across the strip chart by the x-axis or by selecting the light blue section of the Log Controller bar with the cursor.

The following cursor appears.

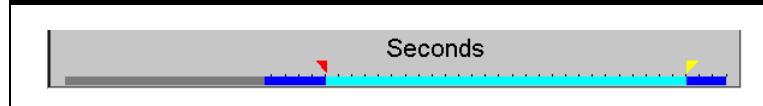


Move the cursor along the bar to the interval you want to view.

The range of sampled data displayed in the strip chart can be manipulated. Below the x-axis, a set of colours appear to indicate what range of sampled data are in the strip chart.

Increase or decrease the range of sampled data and scroll the strip chart over a range of recorded strip chart data. Depending on the amount of data already recorded in the strip chart view, the Log Controller bar, shown in the figure below, appears below the x-axis.

Figure 11.14



Each colour (from left to right) in the Log Controller bar is described in the following table:

Colour	Definition
Gray Bar	There is no data in the strip chart.
Dark Blue Bar	Strip chart data is recorded.
Red Marker	Indicates where the first data displayed in the strip chart is located in the overall data set. Expand the range of display by "dragging" the red marker to the left (away from the yellow marker) and decrease the displayed range by dragging the red marker right (towards the yellow marker).
Light Blue Bar	Indicates where the data displayed on the strip chart is located in the overall data set.
Yellow Marker	Indicates where the displayed data ends. Expand the displayed range by "dragging" the yellow marker to the right (away from the red marker) and decrease the displayed range of data by dragging the yellow marker left (towards the red marker).

Creating Interval Markers

Interval markers are used to measure variables at certain instances in the strip chart. The strip chart variable value appears next to the intersecting point of the interval marker and the strip chart variable curve. Up to four interval markers can be added to the strip chart.

1. Ensure the most recent data appears on the strip chart. The light blue portion of the Log Controller Bar should be located to the far right of the x-axis.
2. Place the cursor on the right edge of the strip chart. A left arrow replaces the cursor.



Left Arrow cursor

3. Click and drag the interval marker across the strip chart. Release the mouse button when the interval marker is in the required location.

Zooming in on the Strip Chart

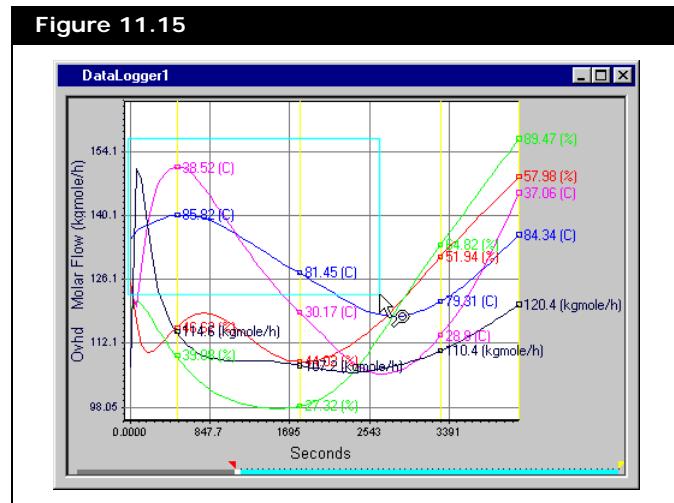


Zoom Strip Chart cursor

Use this procedure to focus or zoom in on an area of the strip chart.

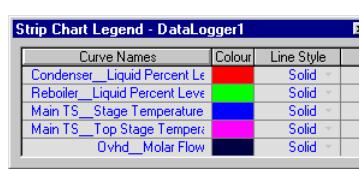
1. Press and hold the **SHIFT** key and press and hold the right mouse button until the cursor changes to a magnifying glass.
2. Drag the cursor until a box encompasses an area in the strip chart.
3. Release the mouse button and the **SHIFT** key. The y-axis scaling changes to reflect the zoom selection.

Figure 11.15



Object Inspect Menu

The following commands appear in the strip chart Object Inspect menu.

Option	Description
Graph Control	Customize the appearance of the strip chart and modify curve and axis parameters. For more information, see the Graph Control section.
Legend	<p>Opens the Legend view.</p>  <p>The Legend view displays all the Curve Names, Colours, and Line Styles associated with the variables in the strip chart.</p> <ul style="list-style-type: none"> • Modify the curve name by clicking in the Curve Name field of the appropriate variable and typing in a new name. • Modify the colour by double-clicking in the Colour field of the appropriate variable and selecting the new colour from the colour palette. • Modify the line style by clicking in the Line Style field of the appropriate variable and selecting a new line style from the drop-down list.
Select Curve	Select a curve on the strip chart.
Auto Scale Curve	The bounds for the y-axis of the selected curve are automatically set.
Copy to Clipboard	Copies the selected view/information to the clipboard for storage and later use. Also, modifies the percentage of the object's size being stored.
Print Plot	Prints the strip chart as it appears on the screen.
Print Setup	Opens the Print Setup view so you can modify any print options associated with printing the strip chart.

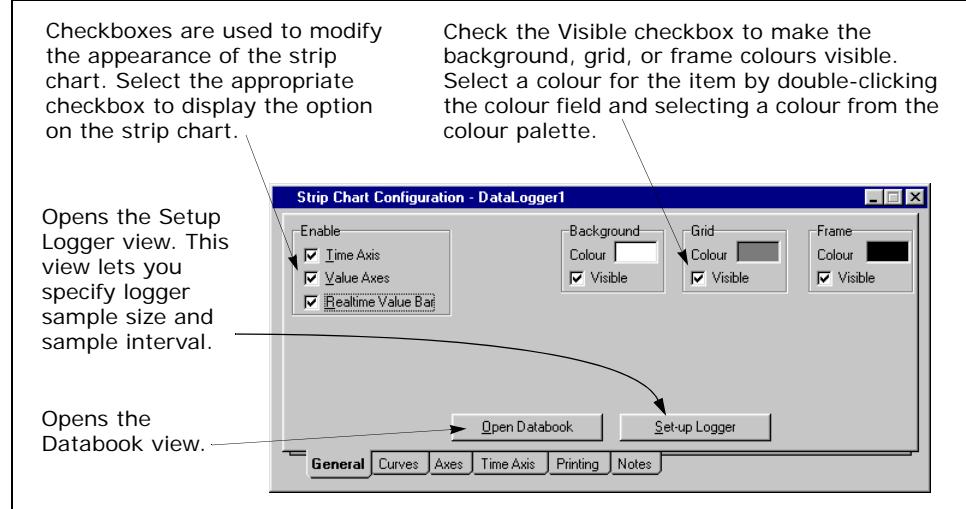
Graph Control

To open the Strip Chart Configuration view, click the Graph Control command in the strip chart Object Inspect menu. Use this view to modify the characteristics of the strip chart.

General Tab

Use the General tab to format the appearance of the strip chart.

Figure 11.16

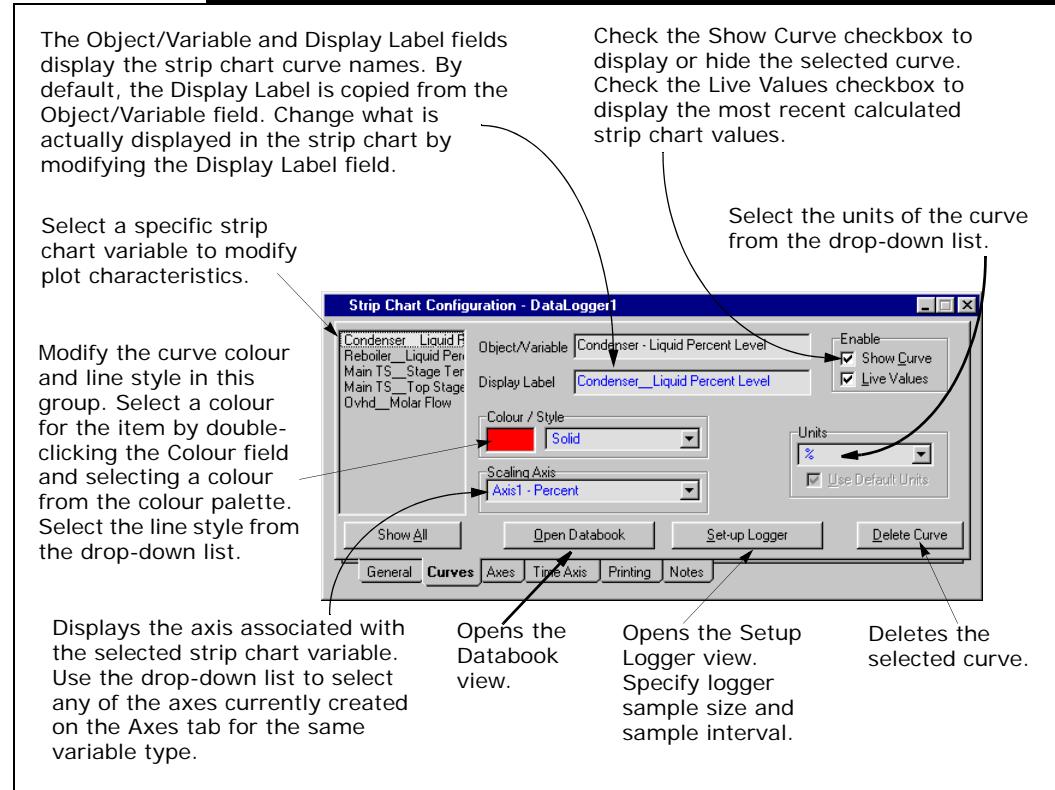


Curves Tab

The Curves tab is used to modify the appearance of individual curves in the strip chart. You can also modify how strip chart variables, variable

titles, and units appear on this tab.

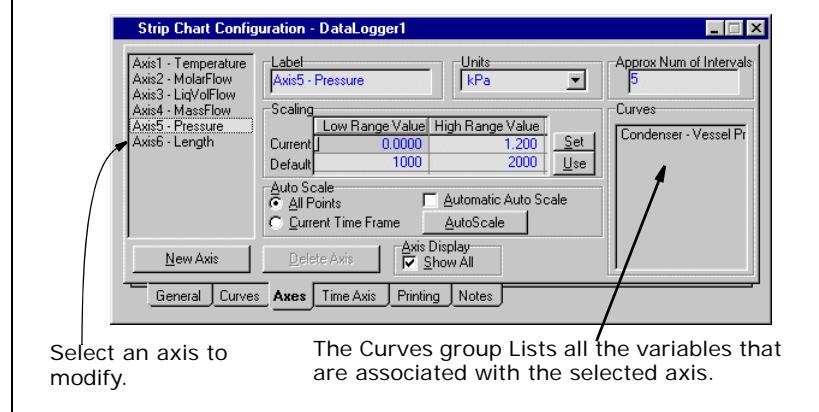
Figure 11.17



Axes Tab

The Axes tab allows you to create, modify, and delete the y-axes.

Figure 11.18



Object	Description
Label field	Allows you to change the label name of the selected vertical axis (y-axis) in the strip chart.
Units field	Allows you to change the unit label of the selected vertical axis by using the drop-down list.
Approx Num of Intervals field	Allows you to set the number of intervals on the vertical axis.
Scaling group	The Scaling group contains the following objects: <ul style="list-style-type: none"> Current row. Displays the actual high and low range values of the axis. Default row. Displays the UniSim Design default high and low range values of the axis. Set button. Allows you to copy the ranges from the Current row into the Default row. Use button. Allows you to copy the default ranges from the Default row into the Current row.
Auto Scale group	This group contains the following objects: <ul style="list-style-type: none"> All Points radio button. Allows you to automatically adjust the scale to best show all of the data points available. Current Time Frame radio button. Allows you to automatically adjust the scale to best show all the data points available within the current time frame. Automatic Auto Scale checkbox. When you check the Automatic Auto Scale checkbox, UniSim Design automatically calculates the most suitable scale setting for all the axes in the strip chart. Uncheck to allow user-specified scaling. AutoScale button. Automatically adjust the high and low range values for all axes. The AutoScale button is automatically disabled when the Automatic Auto Scale checkbox is active.
New Axis button	Allows you to create a new axis in the strip chart.
Delete Axis button	Allows you to delete the selected axis created by the user. The UniSim Design default axes cannot be deleted.
Axis Display group	Check the Show All checkbox to display all the vertical axes with individual scale settings in the strip chart. Uncheck to display only one selected axis.

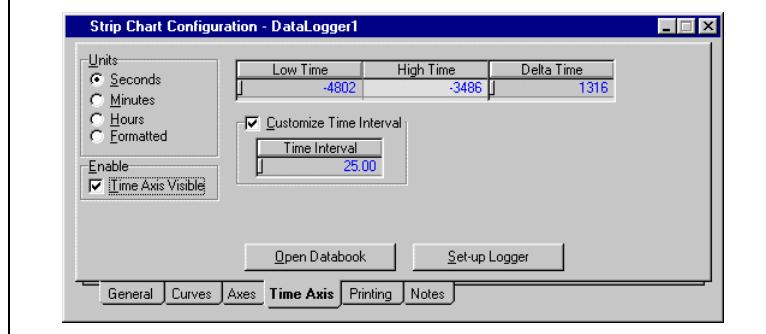
The Automatic Auto Scale checkbox is automatically unchecked when you move the curves, adjust the grid, zoom in/out, or scales a single curve in the strip chart.

Time Axis Tab

The Time Axis tab allows you to modify the display and range of the x-

axis.

Figure 11.19



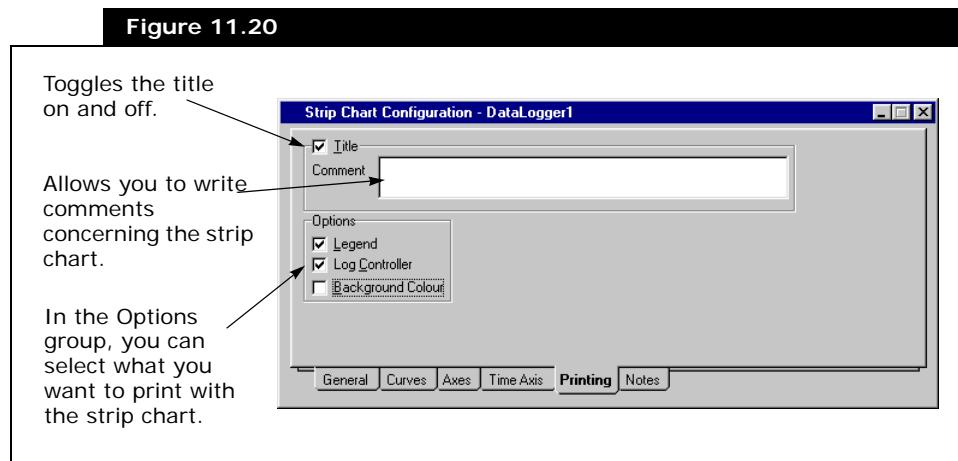
The following table lists and describes the objects on the Time Axis tab.

Enter time using the following format:
HHH:MM:SS.S

Object	Description
Units group	This group contains four radio buttons: <ul style="list-style-type: none"> Seconds. Displays the time in seconds on the x-axis. Minutes. Displays the time in minutes on the x-axis. Hours. Select to have the time displayed in hours on the x-axis. Formatted. Select to have the time displayed in standard time display (i.e. hour:minute:second.second).
Low Time field	Allows you to enter the low range value of the time shown on the x-axis on the strip chart.
High Time field	Allows you to enter the high range value of the time shown on the x-axis on the strip chart.
Delta Time field	Allows you to enter the difference between the high and low range value of time. If the Low Time and High Time fields are specified, the Delta Time field shows the time difference.
Customize Time Interval checkbox	When you check the Customize Time Interval checkbox, the Time Interval field is enabled. In the Time Interval field, you can specify the size of each time interval on the x-axis.
Enable group	Allows you to show the time axis label on the strip chart. When you uncheck the Time Axis Visible checkbox, the time axis label is removed from the strip chart.
Open Databook button	Opens the Databook view.
Set-up Logger button	Opens the Setup Logger view which allows you to specify the logger sample size and the sample interval.

Printing Tab

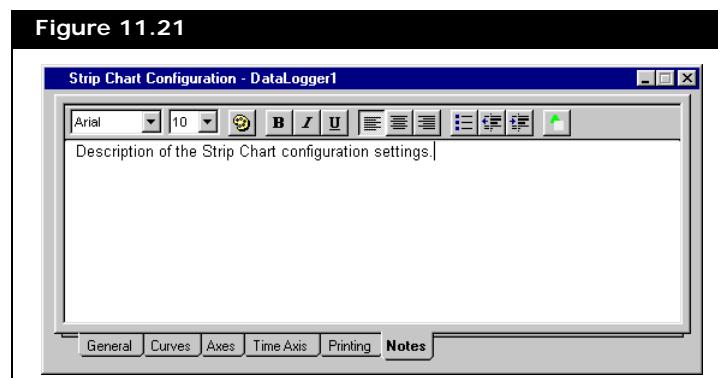
The Printing tab is used to control the printed output of the strip chart.



Notes Tab

To see all notes entered in the simulation case, refer to [Section 7.19 - Notes Manager](#).

Enter notes or comments about the strip chart in the notes tab to let others know what the chart displays.



Viewing Historical Data

1. From the list of available strip charts, select the strip chart

you want to view.

2. Click the **Historical** button. The Historical Data view appears.

Figure 11.22

Time [seconds]	2 - Actual Gas Flow [ACT_m3/h]	3 - Actual Liquid Flow [m3/s]
0.000000	888.995	3.75235e-004
20.0000	97.0687	5.58438e-005
40.0000	97.6523	5.43427e-005
60.0000	97.6518	5.41269e-005
80.0000	97.6503	5.40999e-005
100.0000	97.6504	5.40965e-005
120.0000	97.6506	5.40962e-005

Ascending Order [Save To .CSV File...](#) [Save To .DMP File...](#)

Resize the Historical Data view to see more information.

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 logger size and sample interval values specified in the Logger Set-Up view. All data in the Historical Data view are displayed in ascending order by default (the Ascending checkbox is checked automatically). You can uncheck the Ascending checkbox to display the data in descending order.

Exporting Historical Data

1. Click either the **Save To .CSV File** or **Save To .DMP File** button. The Save File view appears.
2. Specify the name and location of your history file.
3. Click **Save**.

Viewing Current Data

1. From the list of available strip charts, select the strip chart you want to view.
2. Click the **Current** button. The Current Data view appears.

Figure 11.23

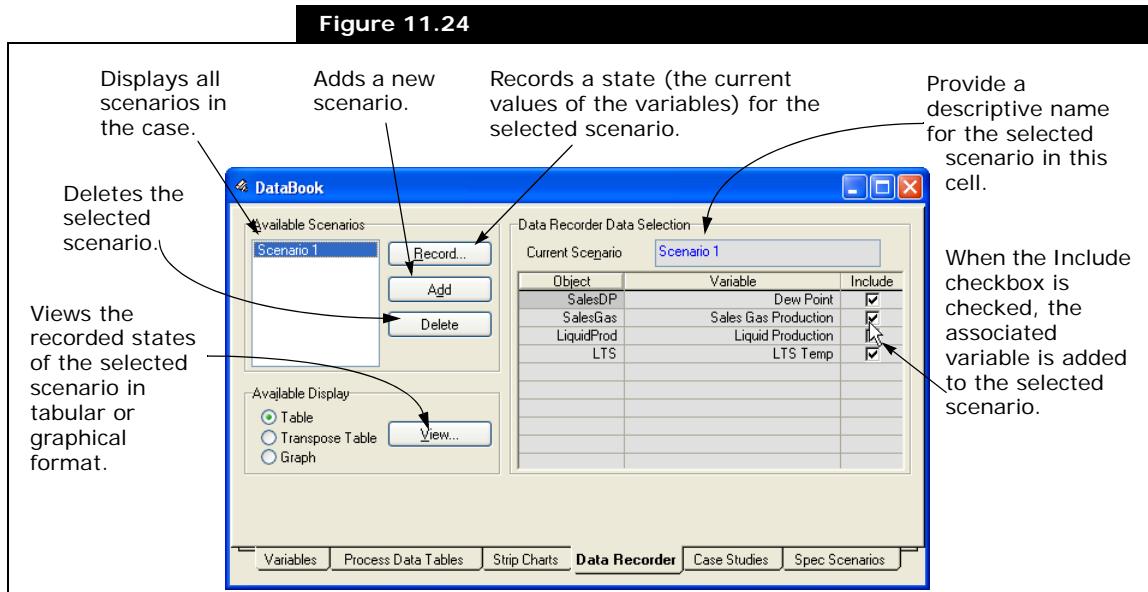
Description	Value	Units	Def
Condenser - Liquid Percer	<empty>	%	<input checked="" type="checkbox"/>
Reboiler - Liquid Percent L	<empty>	%	<input checked="" type="checkbox"/>
Main TS - Stage Tempera	83.98	C	<input checked="" type="checkbox"/>
Main TS - Top Stage Tem	31.48	C	<input checked="" type="checkbox"/>
Dvhd - Molar Flow	110.7	kgmole/h	<input checked="" type="checkbox"/>

11.7.4 Data Recorder

Variables for the data recorder are selected on the Variables tab of the Databook.

The Data Recorder tab lets you store snapshots of your process by grouping key process variables into different scenarios. You can manipulate the process in the current case and then record the results for the variables you are monitoring. Each scenario created can have an unlimited number of snapshots, called States.

Figure 11.24



Adding a Scenario

You can create an unlimited number of scenarios.

Variables can be included in more than one scenario.

1. Click the **Add** button.

A scenario with the default name Scenario 1 appears in the list of available scenarios. If required, type a new name in the **Current Scenario** field.

2. From the list of available variables, check the **Include** checkbox for each process variable you want to add to the scenario.

Recording States

When you make changes to your process, you can record the values of the key variables contained in the scenario.

1. From the list of available scenarios, select the scenario you want to use.
2. Click the **Record** button.

The New Solved State view appears. The initial default name for the new state is State1. Each time you click the **Record** button, the



New Solved State view

integer value in the state name incrementally increases. If required, modify the State name directly in the **Name for New State** field.

If adding a variable to a Scenario after States are recorded, values for the new variable do not appear in the previously recorded States.

- Click the **OK** button. The key process variables of the selected Scenario are recorded.

Repeat these steps each time you want to record the process variables in the scenario. You can record an unlimited number of states for a given scenario.

Viewing a Scenario

If you create more than one scenario, the scenarios are grouped in a Notebook format.

States within a scenario can be viewed in either tabular or graphical format.

- From the list of available scenarios, select the scenario you want to view.
- Select either the **Table** or **Graph** radio button.
- Click the **View** button. The Data Recorder view appears.

Figure 11.25

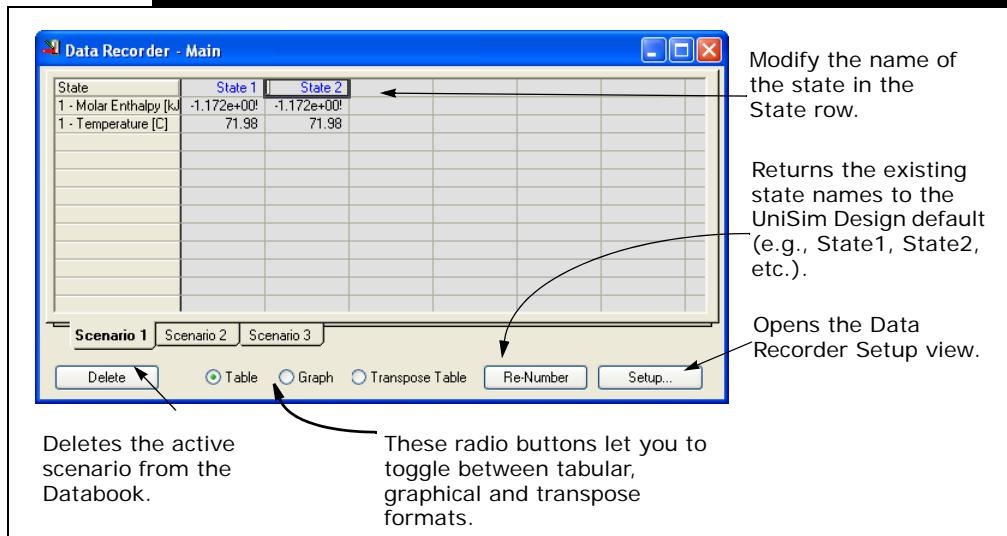
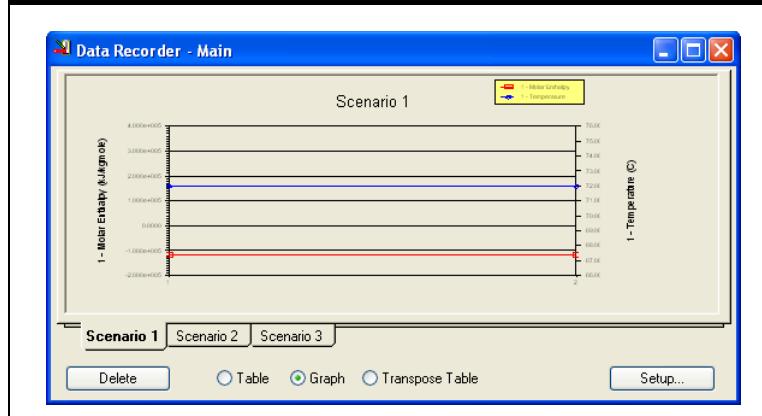


Figure 11.26



Refer to [Section 10.4 - Graph Control](#) for information about customizing plots.

Customize the scenario plot by accessing the Graph Control property view. Right-click anywhere on the plot area and select the Graph Control command from the Object Inspect menu.

If more than two variables are active in the [Data Recorder Setup](#) view, only the first two are plotted.

Deleting a Scenario

When a scenario is deleted, the attached variables are not deleted from the Databook.

- From the list of available scenarios, select the scenario you want to delete.
- Click the **Delete** button.

You will not be prompted to confirm the deletion of a scenario, so ensure the correct scenario is selected before deleting.

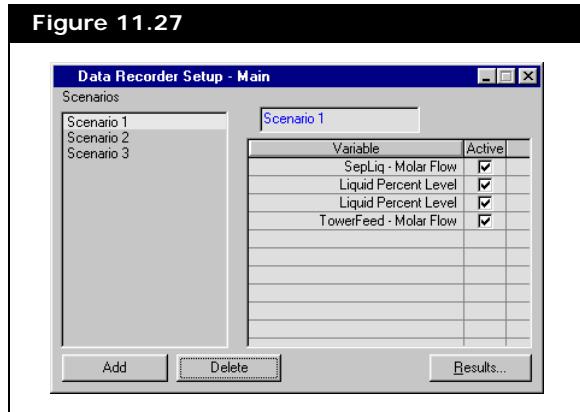
Deleting a State

- From the list of available scenarios, select the scenario you want to view.
- Select either the **Table** radio button or the **View** button. The Data Recorder view appears.
- Click in the column of the state you want to delete.
- Press the **DELETE** key.

You will not be prompted to confirm the deletion of a state, so ensure the correct state is selected before deleting.

Data Recorder Setup

The Data Recorder Setup view provides a list of all scenarios installed in the Databook. From this view, select the variables that appear in the table and on the y-axis of the plot for each scenario. There is a limit of two y-axis variables per plot.



Select the variables you want to display.

1. From the list of available scenarios, select the scenario you want to modify.
2. From the list of variables, check the **Active** checkbox for the variable(s) to be displayed on the plot and in the table.

The following buttons are also available in this view:

Button	Description
Add	Adds a scenario to the Databook.
Delete	Removes the selected scenario from the Databook.
Results	Opens the Data Recorder view to the tab of the selected scenario.

11.7.5 Case Studies

Variables for the case study are selected on the Variables tab of the Databook.

Use the case study tool to monitor the response of key process variables to changes in your steady state process.

1. From the list of variables created on the **Variables** tab, designate the independent and dependent variables for each case study.
2. The independent variables can be input either in Discrete states or as Nested variable ranges. If the Nested input method is selected, each independent variable may be changed by three ways: step size, logarithm step size, or total number of calculation points. Refer to Independent Variable Setup in sub-section Defining a Case Study for more information on the various input methods .

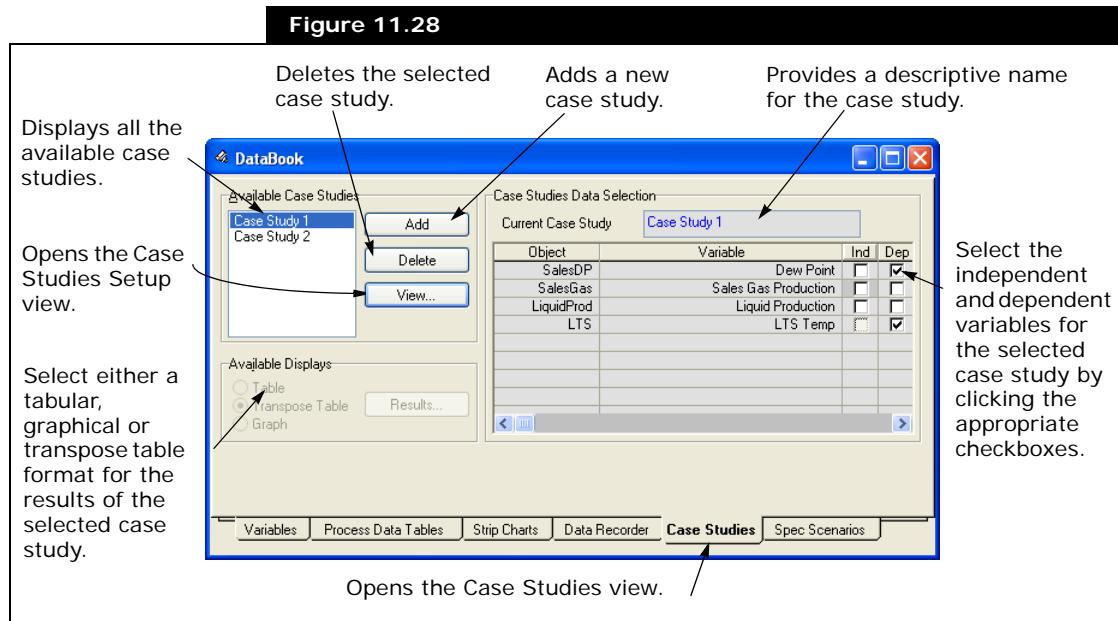
When the Nested method is selected, the independent variables are varied one at a time and with each change, the dependent variables are calculated and a new state is defined. As you define the bounds and step size of the independent variables, the number of states that are calculated are shown.

Since a maximum of two independent variables can be plotted, limit the number of states and minimize solution time by selecting only two independent variables per case study.

After the case study solves, examine the states in a table or view the results in a plot. Although you can select as many variables as you want for a case study, a maximum of three variables can be shown on a plot.

One independent variable and two dependent variables produce a two-dimensional plot while two independent variables and a single dependent variable appear on a three-dimensional graph.

Figure 11.28



Adding a Case Study

1. Click the **Add** button. A scenario with the default name Case Study 1 appears in the list of available case studies. If necessary, type a new name in the **Current Case Study** field.
2. In the Case Studies Data Selection group, check the **Ind** checkbox for the independent variables that UniSim Design varies.
3. Check the **Dep** checkbox for the calculated dependent variables.

Deleting a Case Study

1. From the list of available case studies, select the case study you

Use the same variables in different case studies.

When a case study is deleted, the attached variables are not deleted from the Databook.

want to delete.

2. Click the **Delete** button.

You will not be prompted to confirm the deletion of a case study, so ensure the correct case study is selected before deleting.

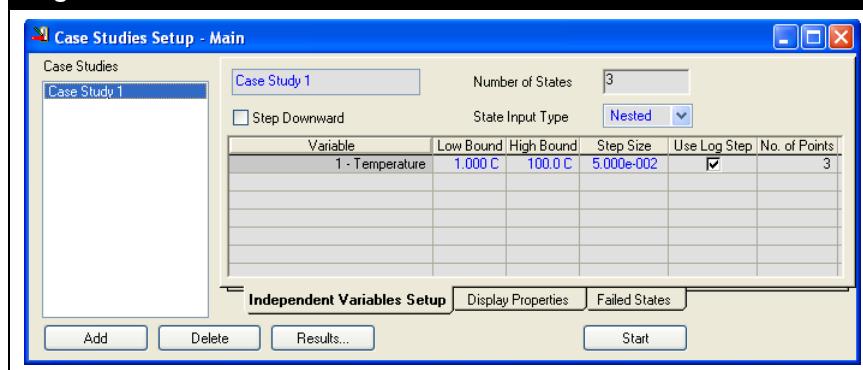
Defining a Case Study

1. From the list of available case studies, select the case study you want to define.
2. Select the independent and dependent variables for the selected case study by checking the corresponding checkboxes.
3. Click the **View** button.

The Case Studies Setup view appears.

You can add and delete case studies from the Case Studies Setup view, but you cannot select the dependent and independent variables.

Figure 11.29

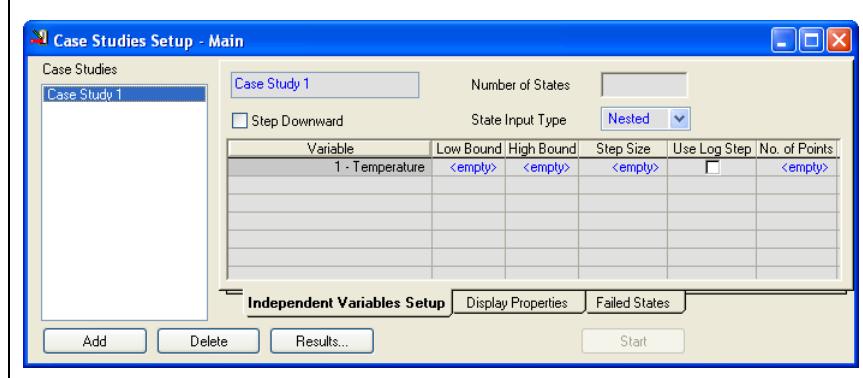


4. Click the **Independent Variables Setup** tab.
5. When **Step Downward** is checked, the case study will run from the high bound and step downward to the low bound.
6. In the State Input Type dropdown box, you can select either Discrete or Nested. When you select Nested as shown in [Figure 11.29](#), you may enter Low Bound, High Bound, Step Size or Number of Points. If you check the Use Log Step checkbox, UniSim Design will use logarithm scale and step size for the chosen independent variable. However, the logarithm scale and step size have their intrinsic limitations and need special handling for some situations. Following are the limitations and implementation details for logarithm scale and step size:
 - Logarithm step size is the power of a multiplier based on 10, i.e., a step size of 1 means a multiplier of 10.
 - UniSim Design internal units are used when calculating the logarithm values, with the only exception being the temperature, for which K is used instead of C.

A maximum of two independent variables can be plotted. If more than two are used in a case study, graphical results are not available.

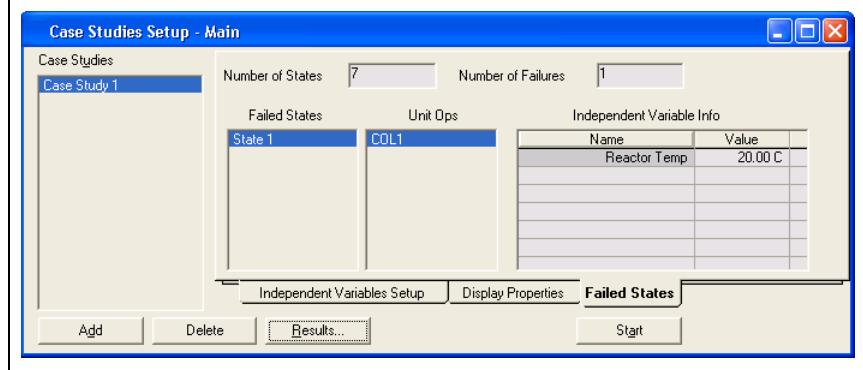
- Negative independent values are not allowed to use logarithm step size except for Temperature where internal unit is used as Kelvin.
 - The minimum value to use logarithm step size is 0, for which the following special handling is implemented: When 0 or any number less than 1.0×10^{-6} (in internal units) is specified as the lower bound, it will be used directly as the first point. However, when calculating the next point, 1.0×10^{-6} is used in place of the specified lower bound.
7. When you select Discrete for State Input Type, the following view appears and you can enter the number of states and the values for the discrete states.

Figure 11.30



8. Click the **Start** button to begin calculations. At any time during the calculations, click the **Stop** button to stop calculating.
9. Click the **Failed States** tab.

Figure 11.31



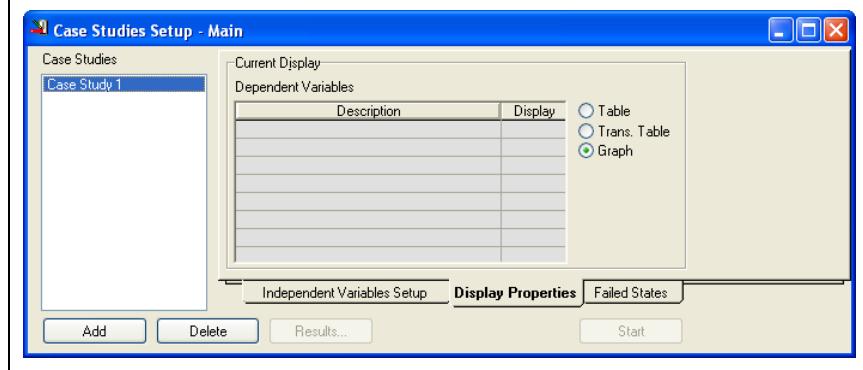
If any of the states could not solve, these states and the unit operation associated with failure appear on this tab.

10. From the list of available failed states, click the failed state to see the value at which the independent variable failed.

11. Click the **Display Properties** tab.

If two independent variables are checked, only the first checked dependent variable appears on the plot. If one independent variable is used, the first two checked dependent variables are shown on the plot.

Figure 11.32



12. From the list of available dependent variables, select the variables for which you want to show results.
13. Click the **Results** button. The Case Studies view appears.

Viewing Case Study Results

The states contained within a case study can be viewed in either tabular or graphical format.

1. From the list of available case studies, select the case study you want to view.
2. Select either the **Table** or **Graph** radio button.
3. Click the **View** button.

The Case Studies view appears.

Figure 11.33

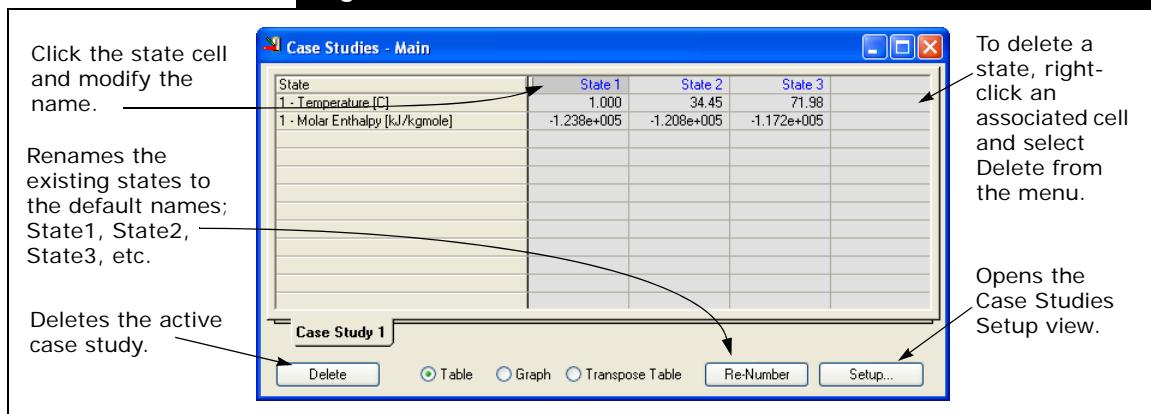
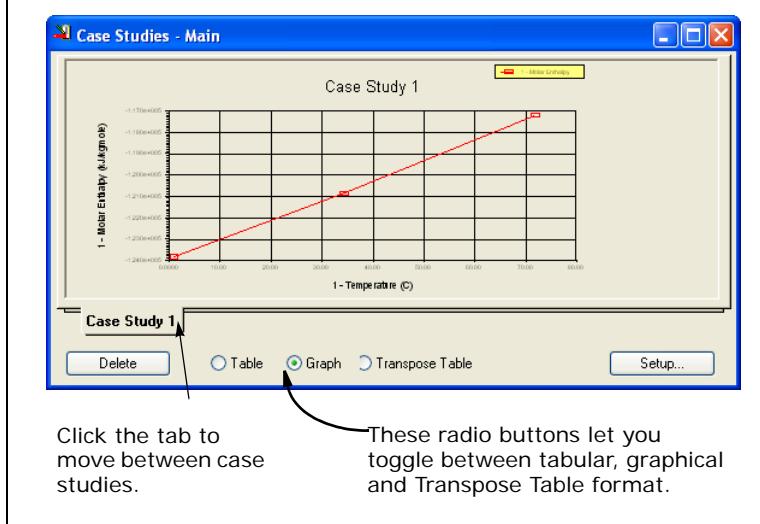


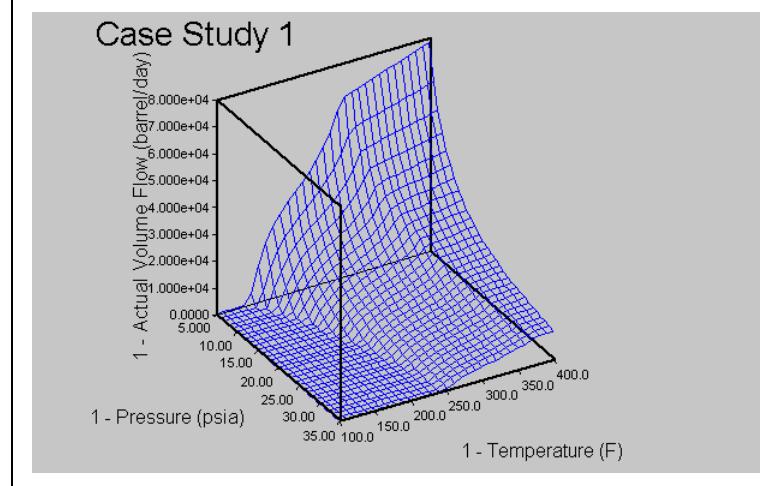
Figure 11.34

Refer to [Section 10.4 - Graph Control](#) for more information about customizing plots.

Customize the scenario plot by accessing the Graph Control property view. Right-click anywhere on the plot area and select the Graph Control command from the Object Inspect menu.

Multi-Dimensional Graphing

When conducting case studies that involve two independent variables, these results are plotted in a three-dimensional graphing environment.

Figure 11.35

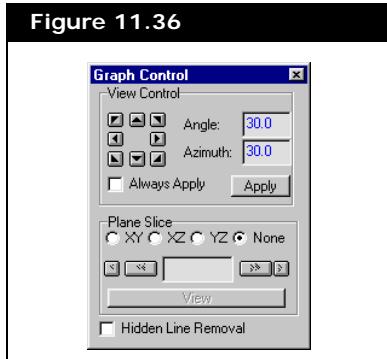
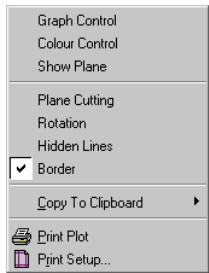
If the case study is run with two independent variables, it automatically

plots three-dimensionally.

The limits of the system allow for only three dimensions. Any more than two independent variables results in no graph being produced.

Multi-Dimensional Graph Control

The options in the Graph Control view can also be found in the Object Inspect menu.



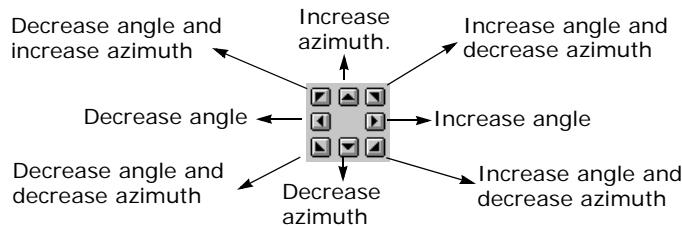
The Azimuth field defines the angle between a horizontal plane and the XY plane of the plot. This means that an azimuth of 0.0 results in a view of the XY plane as a single horizontal line across the screen. The azimuth can be manipulated by clicking in the Azimuth field, entering a value, and then clicking the Apply button.

The Angle field defines the angle between the vertical and the XZ (YZ) plane of the plot. Increasing the angle causes the graph to rotate counter-clockwise. The angle can be manipulated by clicking in the Angle field, entering a value, and clicking the Apply button.

Alternatively, the angle and azimuth can be manipulated by pressing the arrow buttons of the keypad in the View Control group. The action

of each arrow button is explained in the figure below.

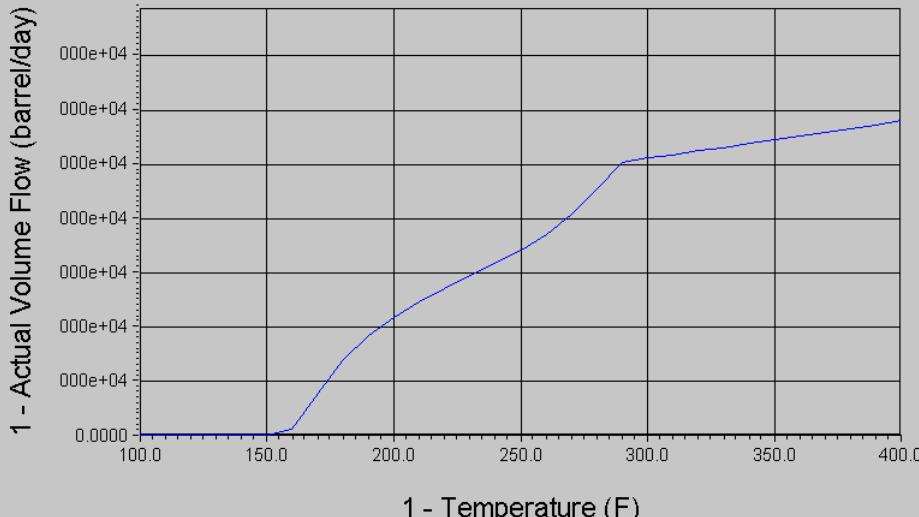
Figure 11.37



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. The two-dimensional plot can be examined by clicking the View button.

A sample plot using the plane slice method appears in the figure below:

Figure 11.38



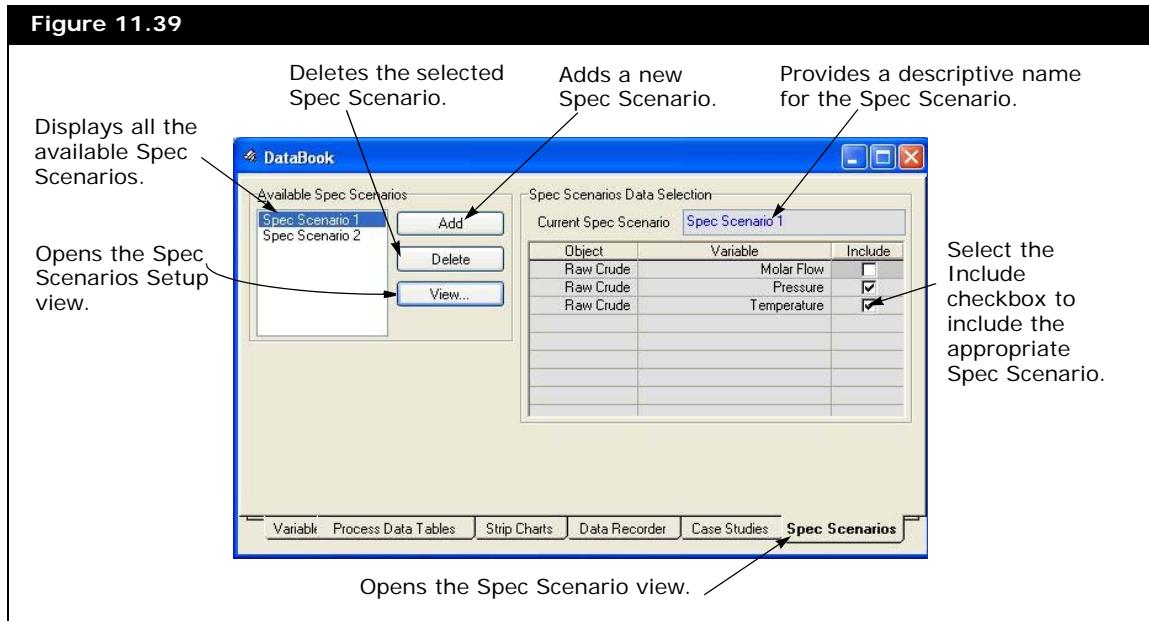
The planar slice can be moved through the plot by pressing the right and left arrow buttons located within the Plane Slice group.

11.7.6 Spec Scenarios

The Spec Scenarios tab lets you enter and then switch between different sets of specifications. You can add any number of specification scenarios and each scenario can contain multiple states. Each state provides specified values for the process. When each state

is applied, the values in that state are used to calculate the flowsheet. If you want to switch which variables are specified in some of the states, simply add all of the potential variables to the scenario and then leave unspecified values empty. Variables for the spec scenarios are selected on the Variables tab of the databook.

Figure 11.39



Use the same variables in different spec scenarios.

When a Spec Scenario is deleted, the attached variables are not deleted from the Databook.

Adding a Spec Scenario

1. Click the **Add** button. A scenario with the default name Spec Scenario 1 appears in the list of available Spec Scenarios. If necessary, type a new name in the **Current Spec Scenario** field.
2. In the Spec Scenarios Data Selection group, check the **Include** checkbox for the variables that UniSim Design varies.

Deleting a Spec Scenarios

1. From the list of available Spec Scenarios, select the spec scenario you want to delete.
2. Click the **Delete** button.

You will not be prompted to confirm the deletion of a spec scenario, so ensure the correct spec scenario is selected before deleting.

Viewing Spec Scenario Results

1. From the list of available Spec Scenarios, select the Spec Scenario

A tab is displayed for each Spec Scenario in your databook.

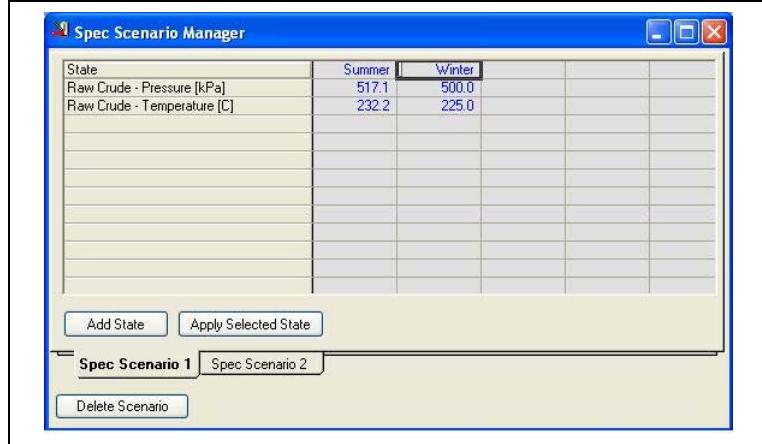
To apply a state to the flowsheet, click the **Apply Selected State** button.

you want to view.

2. Click the **View** button.

The Spec Scenario view appears.

Figure 11.40

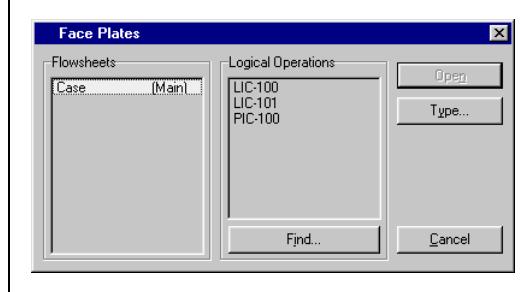


11.8 Face Plates

Refer to [Section 12.13 - Controller Face Plate](#) in the [UniSim Design Operations Guide](#) for more information.

The face plate provides all pertinent information about a controller. To access the Face Plates Manager, select the Face Plates command from the Tools menu, or press the **CTRL F** hot key.

Figure 11.41



The Face Plate Manager lets you quickly search all available flowsheets in the case and open the faceplate of the controller you want.

Opening a Face Plate

1. From the list of available flowsheets, select the flowsheet you want to search.
2. From the list of logical operations, select the controller for which you want to view the face plate.

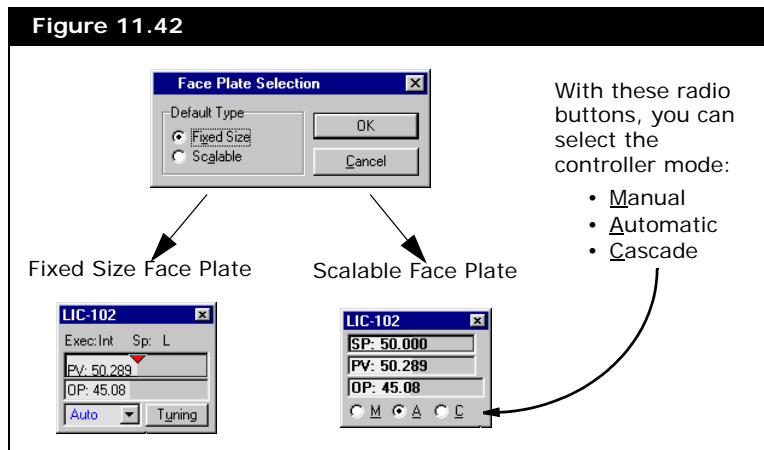
- Click the **Open** button to display the face plate for that controller.

OR

- If you know the name of the controller, but not the location, click the **Find** button. The Find Object view appears.
- Type in the name of the controller, then click **OK**. The controller property view appears, giving you access to all the controller parameters and the controllers face plate.

Face Plate Types

Change the appearance of all face plates in the flowsheet by clicking the Type button. Two types of face plates available are Fixed Size (the default) and Scalable. Both types appear in the figure below.



When the scalable face plate type is selected, you can change the face plate font by clicking the Set Font button in the Face Plate Manager.

11.9 Dynamics Assistant

For more information about the Dynamics Assistant, see [Section 2.2 - Dynamics Assistant](#) in the UniSim Design Dynamic Modeling Guide.

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

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

11.10 Control Manager

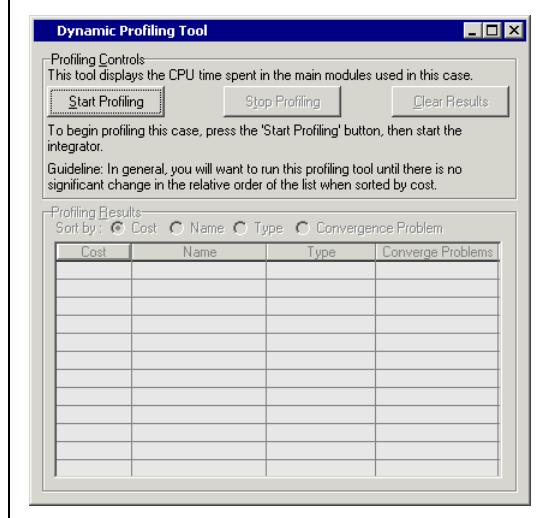
For more information about the Control Manager see [Section 2.6 - Control Manager](#) in the **UniSim Design Dynamic Modeling Guide**.

The Control Manager command opens the Control Manager view. This view contains a summary of the PID Controllers and MPC Controllers contained within the current simulation.

11.11 Dynamic Profiling Tool

The Dynamic Profiling Tool logs the CPU time spent in the main modules of a case during dynamic simulation. Therefore, it allows you to compare the speed of the modules within the case.

Figure 11.43



There are two groups in the Dynamic Profiling Tool view:

- Profiling Controls
- Profiling Results

Profiling Controls

The buttons found in the Profiling Controls group are:

- **Start Profiling.** The profiling tool begins to record data from an active case.
- **Stop Profiling.** The profiling tools stops recording data from a case.
- **Clear Results.** This clears the results that are currently in the Profiling Results group.

Profiling Results

The Profiling Results group displays a table with four categories and their corresponding radio buttons. The table displays the following information:

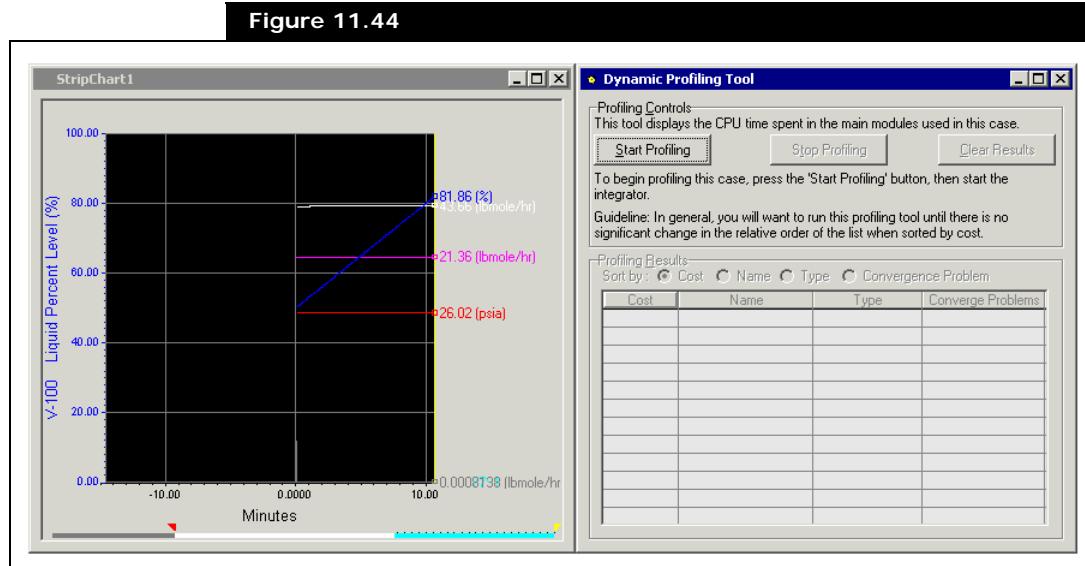
- **Cost.** The CPU cost of an object for a certain integration time. The CPS time associated with flashes is attributed to the object and not the pressure flow solver module.
- **Name.** The name you have given to the object in your case.
- **Type.** The type of object in your case. (e.g., valve, separator, feeder block, etc.)
- **Converge Problems.** The status of the pressure flow solver convergence problems associated with a module. The checkbox is checked if the module requires four or more pressure flow solver iterations during the integration period. Four iterations represents a high iteration count.

11.11.1 Profiling a Case

To profile a case:

1. From the **Tools** menu, select **Dynamic Profiling Tool**. The Dynamic Profiling Tool view appears.
2. Ensure that the StripChart view and Dynamic Profiling Tool view are both visible.

Figure 11.44



3. Click the **Start Profiling** button.

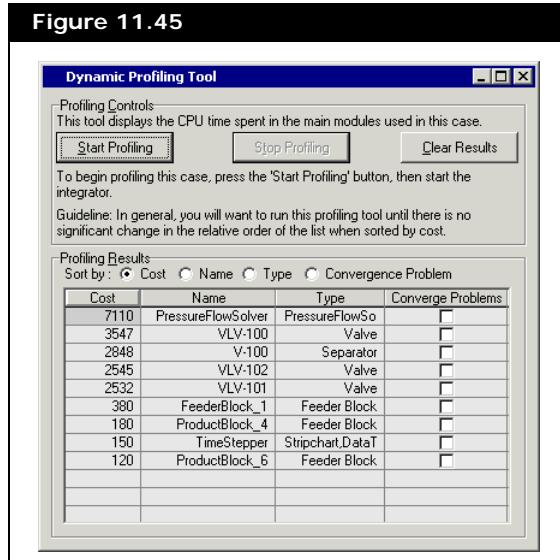


Integrator icons
Green = Active
Red = Holding

- Click the **Integrator Active** icon.

You should run the profiling tool until there is no significant change in the order of the results when it is sorted by cost.

- When you want to stop logging the profiling data, click the **Integrator Holding** icon.
- Click the **Stop Profiling** button and the results are displayed in the table in the Profiling Results group.



If you start the profiling tool again after receiving results, it will ask if you want to clear your current results or add to them. Click the Yes button if you want to clear the results, or click the No button if you want the results to accumulate.

- You can sort the results by Cost, Name, Type, or Convergence Problems by clicking on the corresponding radio button.

The information in the Converge Problems column of the profiling results is most useful to a developer and not the general user. The checkbox is checked if a module requires four or more pressure flow solver iterations during the integration period. A high convergence count may be associated with a particular operating region, or stiff or non-linear equations associated with that module. This information is useful to a developer in debugging a particular problem with a case that could be improved via code changes. If a user has written their own extension and they are having problems (e.g., case is running really slow) the convergence problems information may direct them to look at certain pressure flow equations in their extension.

The Profiling Tool can be useful in a variety of ways, especially when you are trying to investigate why a dynamics case is running particularly slow. When you compare the profiling results from two runs of the same case, you can point to problems with a unit operation/module if it is particularly expensive in one of the runs.

As well, you can compare the profiling results by object type. For example, if one valve is much more expensive than the rest, then that valve deserves some investigation.

11.12 Snapshot Manager

The Snapshot Manager allows you to create a snapshot of the simulation during dynamics modeling. A snapshot is a file that contains the state of the simulation at a particular time when the integrator is still running. Snapshot files allow you to trace back to a specific point in the simulation and review the results calculated at that time. This is especially useful for Operation Training System (OTS) applications, where the instructor can routinely save the state of the simulation at various times during a training exercise for an extended period of time. Students or trainees can then study and compare the result from the snapshot files.

The Snapshot Manager consists of three tabs:

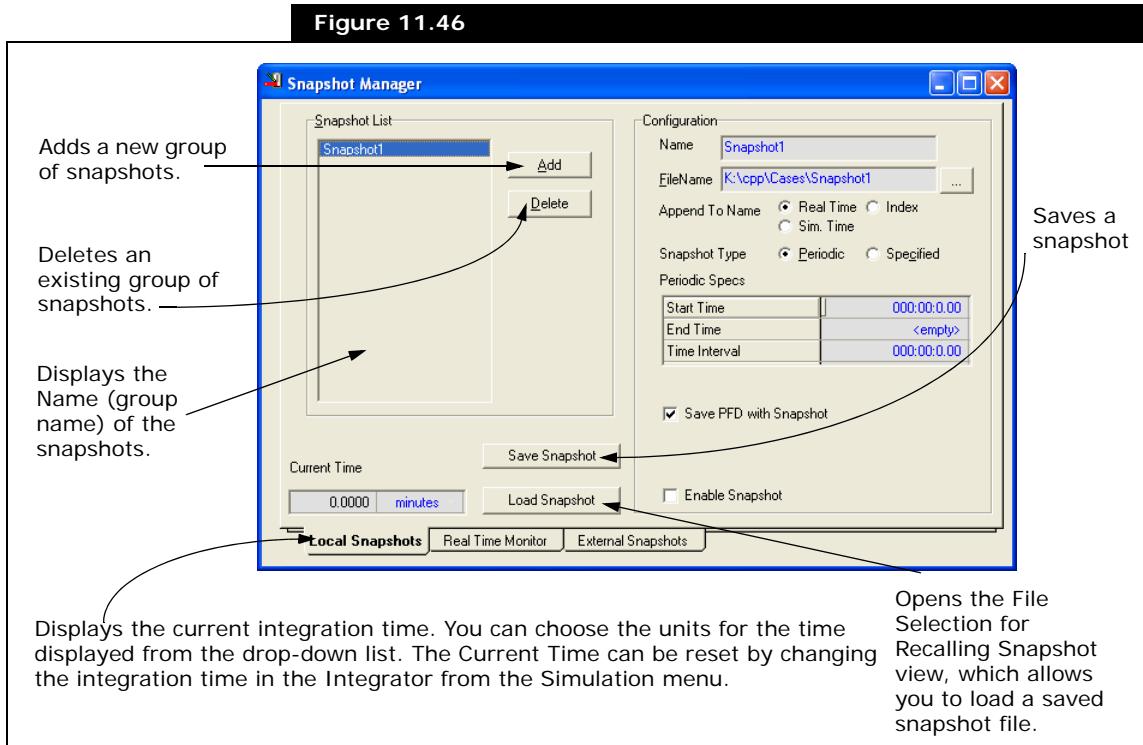
- Local Snapshots
- Real Time Monitor
- External Snapshots

11.12.1 Local Snapshots Tab

The Local Snapshots tab allows you to save, load, and configure a

snapshot.

Figure 11.46



Refer to [Chapter 7.6 - Integrator](#) for more information on the Integrator.

The Configuration group is used to specify how you want the snapshot to be taken. The following table describes each parameter in the Configuration group:

Ellipsis icon

Object	Description
Name	Displays the group name of the snapshots in the Snapshot List.
FileName	Displays the root directory path and the generic name of the snapshots. The root path must contain the generic name of the snapshots. By default, the generic name is set to be the same as the Name (group name) of the snapshots. Click the Ellipsis icon to change the directory or file name. The File Selection for Saving Snapshot view appears.
Append to Name	Consists two radio buttons: <ul style="list-style-type: none"> Real Time. Attaches the current date and real time to the FileName of the snapshot (e.g., snapshot1_2002-12-24_10-20-15.usp). Sim Time. Attaches the current date and simulation time to the FileName of the snapshot (e.g., snapshot1_2002-12-24_10-20-15.usp). Index. Labels the snapshot in chronological order by attaching an index number at the end of the FileName. (e.g., snapshot1_1.usp).

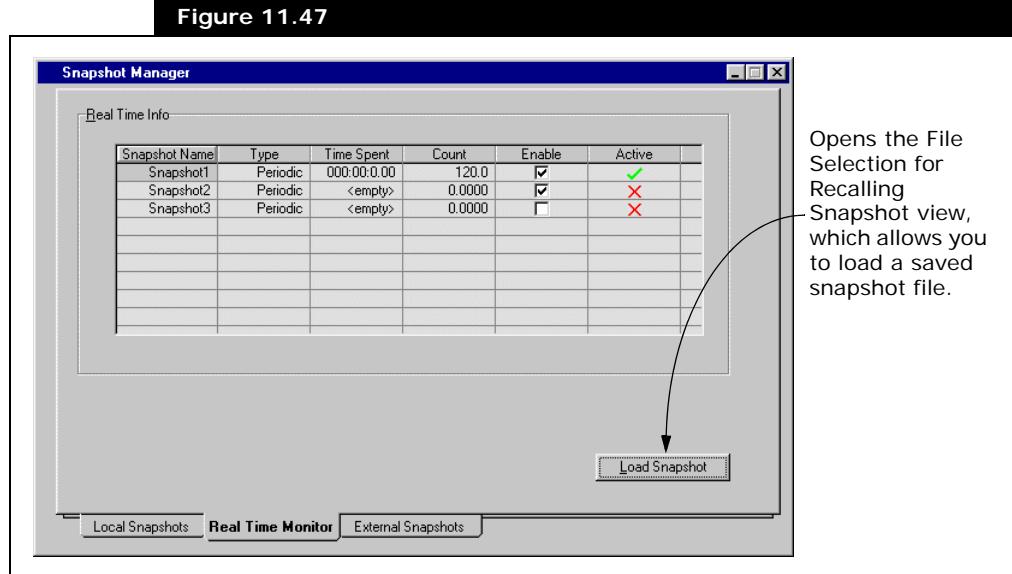
Object	Description
Snapshot Type	<p>Consists of two radio buttons:</p> <ul style="list-style-type: none"> Periodic. The Periodic Specs table appears upon activating the Periodic radio button. In the table, you can set the snapshot capturing parameters by specifying the Start Time, End Time, and Time Interval. Specified. A table should appear upon activating the Specified radio button. In the Specified Time column, you can specify the time (real time) that you want to take the snapshot. You can choose the unit of input from the drop-down list. UniSim Design automatically converts your input into standard time display. The status of the snapshots are shown in the Processed column. The Specified Time values must be equal or larger than the Current Time value.
Periodic Specs	<p>The table contains three fields:</p> <ul style="list-style-type: none"> Start Time. Allows you to specify the time to take the first snapshot. You can choose the unit of input from the drop-down list. UniSim Design automatically converts your input into standard time display. End Time. Allows you to specify the time to take the last snapshot. You can choose the unit of input from the drop-down list. UniSim Design automatically converts your input into standard time display. The End Time value must be larger than the one of Start Time. If the field is left blank as <empty>, End Time is equivalent to infinity. Time Interval. The time interval between capturing each snapshot. It is the capturing frequency of the snapshots. The minimum value for the Time Interval is 1 second. If the Time Interval field is left blank as <empty>, no snapshots will be taken.
Save PFD with Snapshot	Allows you to save the PFD with the simulation results in one single file.
Save PFD in Single Separate File	Allows you to save the PFD in a separate file without saving it each time as the simulation results are being saved. This speeds up the saving process of the snapshot.
Restore PFD Upon Load if Saved in Separated File	Allows you to restore the PFD upon load if the PFD was saved in a separate file when the snapshot is saved.
Enable Snapshot	Allows you to manually activate the snapshot option according to the specified settings.

11.12.2 Real Time Monitor

The Real Time Monitor gives you an overview of the status of each

group of snapshots.

Figure 11.47



The following table lists and describes the columns in the Real Time Info table.

Object	Description
Snapshot Name	Displays the group name of the snapshots in the Snapshot List, and the Name field on the Local Snapshots tab.
Type	Displays the type (Periodic or Specified) of snapshot you specified in the Configuration group on the Local Snapshots tab.
Time Spent	Displays the time spent on saving the last snapshot.
Count	Displays the number of snapshots taken under the same group.
Enable	When you check the Enable checkbox, the snapshots are taken under the specified conditions in the Local Snapshots tab.
Active	Displays the status of a group of snapshots. A green check indicates that all the specifications are valid and the Snapshot Manager is ready to take the snapshots. A red cross indicates that some of the specifications are not valid or the Periodic Specs have expired.

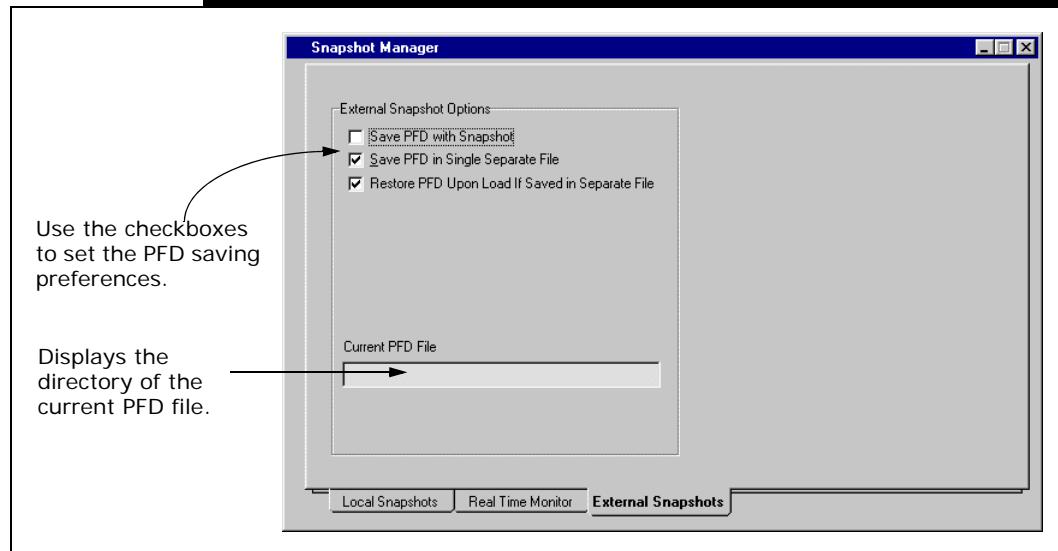
11.12.3 External Snapshots

UniSim Design provides other means to configure and capture a snapshot without using the Snapshot Manager.

The External Snapshots tab allows you to set the PFD saving preferences when you are taking a snapshot outside of the Snapshot Manager. These saving preferences are the same as the first three

checkboxes in the Configuration group on the Local Snapshots tab. You have the options to save the PFD with the simulation results in one file or separate file, and you can choose to have the PFD restored upon loading if the snapshot is saved in a separate file.

Figure 11.48



You can take a snapshot by using one of the following methods:

- Save As option in the File menu
- Event Scheduler
- OLE (Object Linking and Embedding)

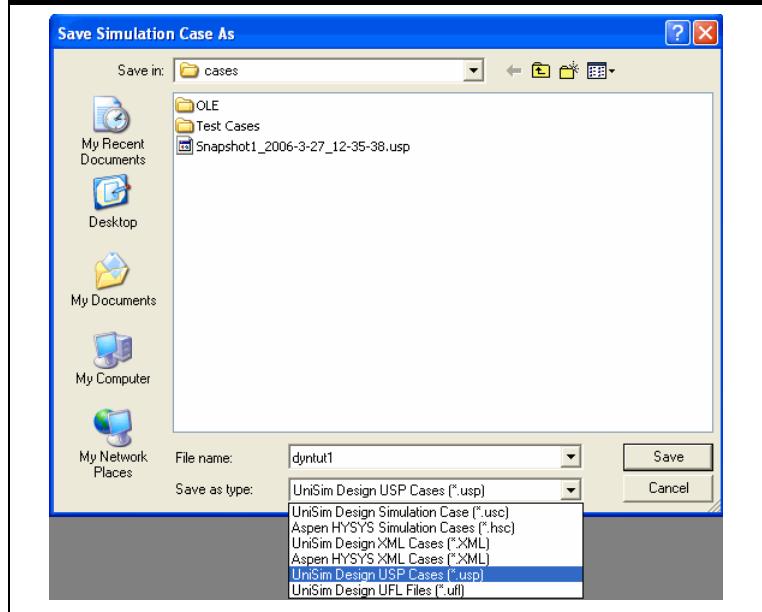
Taking a Snapshot with the Save As Option

The Save As option provides you with the most direct way to take a snapshot. You can manually take a snapshot at any point during the simulation with the integrator running.

From the Save As Type drop-down list, select UniSim Design USP Cases

(*.usp) as shown in the figure below.

Figure 11.49



In the File name field, type the name you want to save the snapshot as. There is no date, time, or index that will be appended to the end of the file name. You can view the last saved PFD location in the Current PFD File field on the External Snapshots tab.

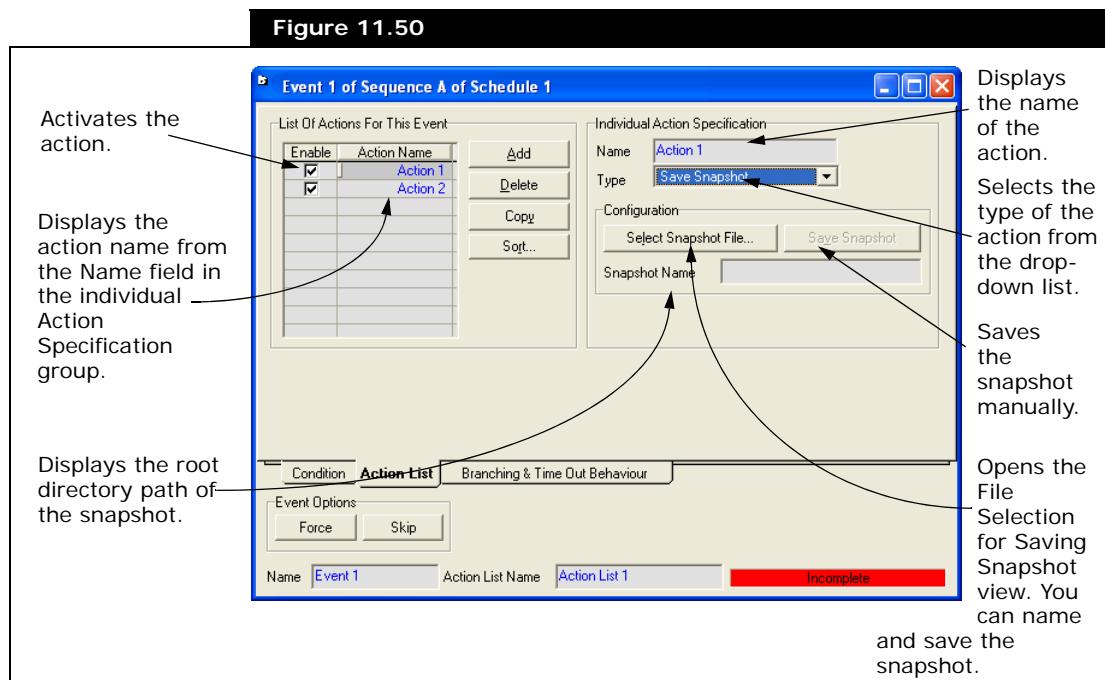
Taking a Snapshot in the Event Scheduler

Refer to [Chapter 7.5 - Event Scheduler](#) for more information on the Event Scheduler.

You can access the Event Scheduler by pressing **CTRL E**.

The Event Scheduler allows you to setup more complicated scenarios for taking a snapshot. With the Event Scheduler, you can set the snapshot capturing conditions to be triggered by a pre-determined simulation time, a logical expression becoming true, or a variable stabilizing to within a given tolerance for a set amount of time. You can also setup the Event Scheduler to take a snapshot when certain operating conditions are reached (i.e., amount of valve opening,

controller output).



Taking a Snapshot with OLE

Refer to [Section 1.2 - Automation & Extensibility](#) in the [UniSim Design Customization Guide](#) for more information on OLE and Automation.

An OLE is a tool that allows you to programmatically interact between two applications. You can program an OLE using Visual Basic to have UniSim Design take the snapshots according to the conditions specified in the user code. This option gives you more flexibility on how to take the snapshots in terms of time and operation conditions. You can even define additional capturing parameters in your code. The PFD saving preferences are still constrained by the specifications on the External Snapshots tab.

11.13 Script Manager

You can use the Script Manager to record all your case interaction including the following:

- Installing streams or operations
- Making connections
- Supplying specifications

The recorded script can be played back later. To access the Script Manager view, select the Script Manager command from the Tools

menu.

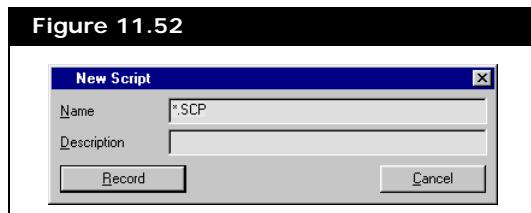


When using the script feature, keep the following in mind:

- Session preferences are not saved in the Script.
- Scripting is always done in UniSim Design internal units.
- Scripting is name specific, so stream and operation names must be identical to those in the case that 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 all the steps in the script can be performed.

11.13.1 Recording a New Script

1. Save your simulation, because the case must be in exactly the same condition for playback of the recorded script.
2. Select the **Script Manager** command from the **Tools** menu. The Script Manager appears.
3. From the list of available directories, select the directory where you want to save the script file.
4. Click the **New** button. The New Script view appears.



5. Type a name and description for the script.
6. Click the **Record** button to start recording. The red Record icon appears in the lower right corner of the Desktop.
7. Perform each task that you want to record.
8. When you finish recording, open the Script Manager and click the **Stop Recording** button.



Record icon

9. Save the case with a different name. If you save the case with the same name, this will prevent you from playing back the script.

11.13.2 Script Playback

For you to run a script, the simulation case must be in the same state as it was prior to the recording.

At any time during the playback, you can stop the script by opening the Script Manager view and clicking the Stop Play button. This stops the script, but does not stop the UniSim Design function that was occurring during playback.



Playback icon

1. Open the case associated with the script.
2. Select the **Script Manager** command from the **Tools** menu. The Script Manager appears.
3. From the list of available directories, select the directory where your script file is located.
4. From the list of available script files, select the script you want to play.
5. Click the **Play** button. The green Playback icon appears in the lower right corner of the Desktop.
6. View the steps of the script playback in the Trace Window.

Refer to [Section 1.3 - Object Status Window/Trace Window](#) for details about the Trace Window.

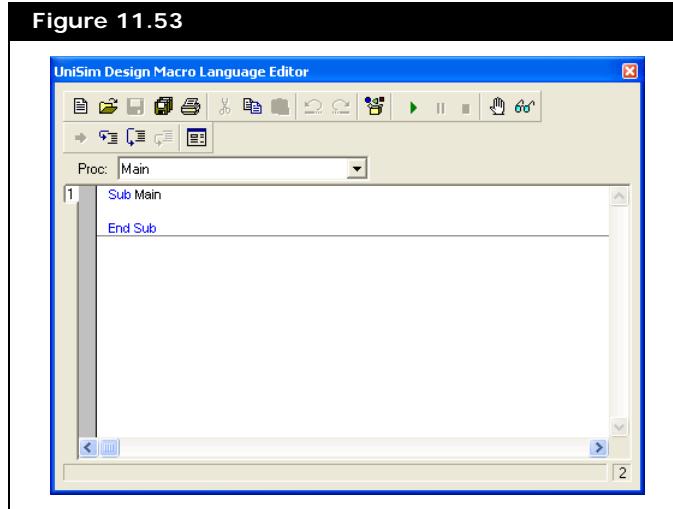
For more information, consult the Online help by clicking **Editor Help** command in the **Help** menu of the UniSim Design Macro Language Editor.

Refer to [Section 2.5 - Example 1: The Macro Language Editor](#) in the **UniSim Design Customization Guide** for an example.

11.14 Macro Language Editor

The UniSim Design 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®.

1. From the **Tools** menu, select **Macro Language Editor**.
2. The UniSim Design Macro Language Editor view appears.

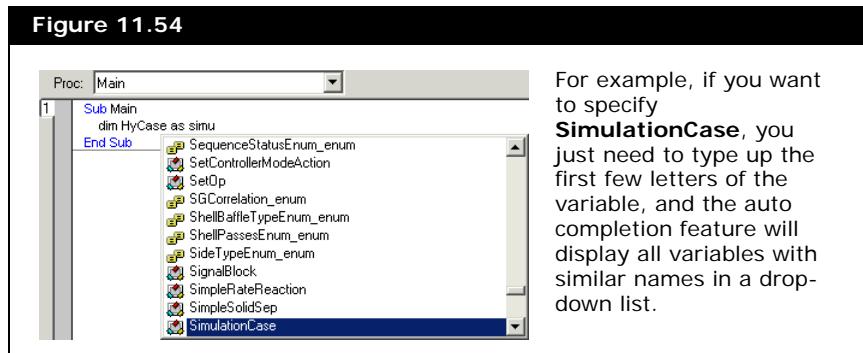


The editor is an interactive design environment for developing, testing and executing WinWrap Basic automation scripts. The editor, which uses a syntax that is very similar to Microsoft®'s Visual Basic, allows you to write code that interacts with UniSim Design.

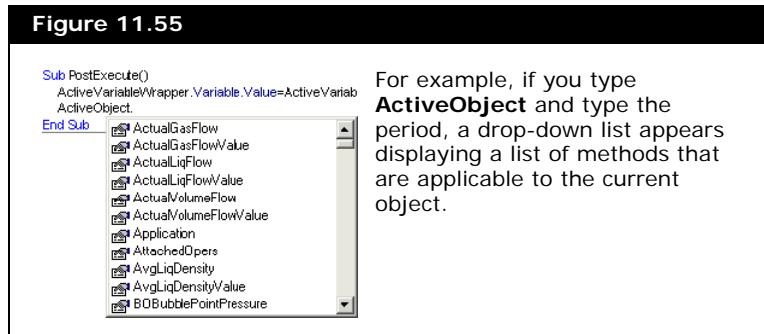
New Features

The Macro Language Editor now has two new features:

- Auto completion feature, which helps you complete the user variable codes and helps you debug the program with flyby evaluation.



- List command feature, which shows you a list of valid methods or properties depending on the context (type of expression) you enter.



11.15 Case Security

Cases can be locked to a password or a password with a security lock device such as a security key.

The Case Security command from the Tools menu enables you to access the Enter Master Password view. This view contains options to lock a UniSim Design case and enable the Runtime Mode.

The Runtime Mode allows you to run a pre-built case with access restricted to certain areas in the case only. This option serves as a security control that allows clients (especially consultants, contractors, and licensors) to deliver a complete UniSim Design model with their end product while protecting their business interests and the intellectual property contained within the product model.

The limitations of a UniSim Design runtime version exist at two levels: case level and application level. The runtime version of the UniSim Design application is compatible only with the runtime version of the UniSim Design cases. Additional licenses are needed to gain full authority to edit the basis and model topology of the runtime case.

11.15.1 Locking a UniSim Design Case

To lock a case, the following three information must be specified:

- master password
- contact name
- company name

1. Select the **Case Security** command from the **Tools** menu. The

Enter Master Password view appears.

Figure 11.56



The password is case sensitive and must be at least six characters in length.

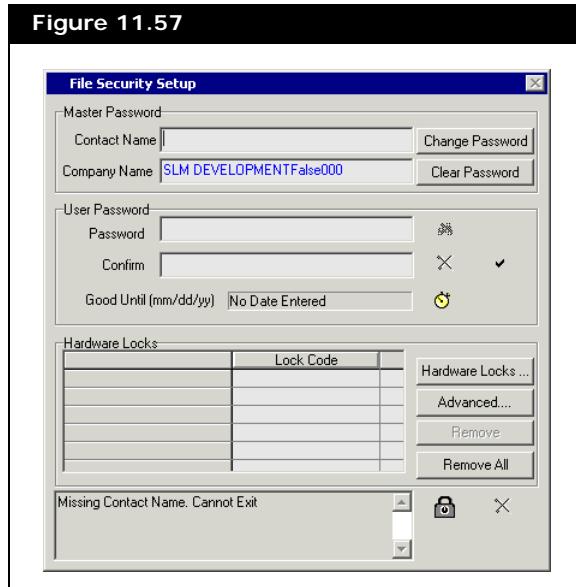


Accept Password icon

2. In the **Case Security** tab, type the password you want to use for the case in the **Password** field.
3. Retype the password in the **Confirm** field.
4. Click the **Accept Password** icon.

The File Security Setup view appears.

Figure 11.57



5. In the Master Password group, enter the name of the contact (typically the name of the user entering the initial password) in the **Contact Name** field.
6. Enter the company name (a UniSim Design default company name is provided, however you can change it) in the **Company Name** field.
7. Check that the **Lock** icon is available in the lower right corner of the File Security Setup view.

If the **Unlock** icon appears, click the **Unlock** icon to activate the **Lock** icon.



Lock icon



Close and Exit icon

8. Click the **Close and Exit** icon to save the case security setting.

11.15.2 Loading a Locked UniSim Design Case

When loading a case that is locked to a password, the following view appears.



Passwords are case sensitive and must be at least six characters in length. You are allowed only three attempts to enter the correct password. If the incorrect password is entered or the correct password is entered but a locking device is not found, the following message appears and the case does not open.



See [Setting a Time Restriction](#) section in the following section for more information about the time restriction date.

If the correct user password was entered, but the time restriction date has expired, then the following message appears:



This message appears only once and uses one attempt of the log on procedure.

11.15.3 File Security Setup

To access the File Security Setup view, select the Case Security command from the Tools menu. One of the following will occur:

- If the case was never locked, the Enter Master Password view appears.



Specify a new password, confirm the password, and click the **Accept Password** icon.

- If the case is locked, the Enter Master Password view appears.



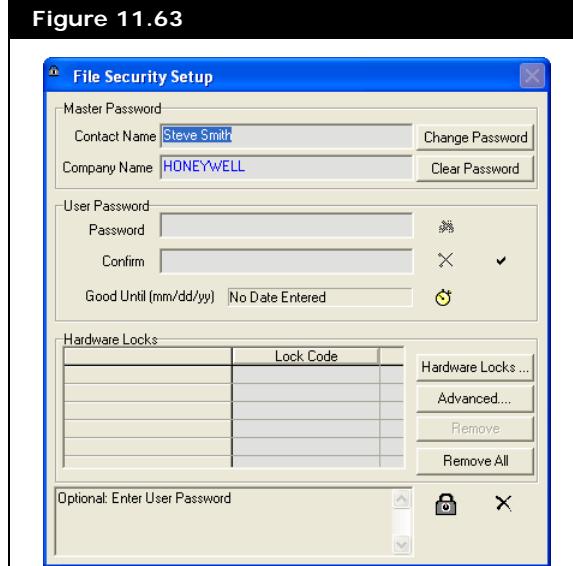
Accept Password icon



Type the correct password and press **ENTER**.

You are allowed only three attempts to enter the correct password. If you exceed the attempt limit, the File Security Setup view does not open and you cannot attempt the access again until the case is reloaded.

The File Security Setup view appears in the figure below.



Object	Icon	Description
Contact Name field		Enables you to specify the name of the person who implemented the password.
Company Name field		Enables you to specify the company name.
Change Password button		Enables you to change the master password.
Clear Password button		Enables you to remove the master password.
Password field		Enables you to specify the user password. The password is case sensitive and must be at least six characters in length.
Confirm field		Enables you to re-enter and confirm the user password.
Good Until field		Displays the expiry date of the user password.
View User Password icon		Enables you to see the user password.
Clear User Password icon		Enables you to remove the user password.
Open Calendar View icon		Enables you to access the Calendar view and select the expiry date for the user password.
Accept User Password icon	<input checked="" type="checkbox"/>	Enables you to save/accept the user password entered in the User Password group.
Hardware Locks table		Displays the status of the selected Hardware locks.
Hardware Locks button		Enables you to access the Scan All Locking Codes view and modify the hardware lock selection.
Advanced button		Enables you to access the Advanced Lock Selection view, and select different hardware lock type.

Object	Icon	Description
Remove button		Enables you to remove the selected hardware lock from the table.
Remove All button		Enables you to remove all the hardware lock in the table.
Case Security is Currently Active icon		Displays the case security status and enables you to change from locked to unlocked case.
Case Security is Currently Disable icon		Displays the case security status and enables you to change from unlocked to locked case.
Close and Exit icon		Enables you to close and save the changes made in the File Security Setup view.

Master Password

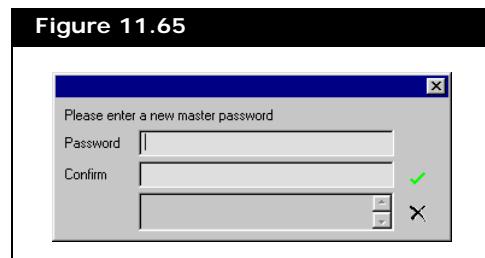
The master password is case sensitive, cannot be less than 6 characters, and cannot be more than 24 characters.

Changing the Master Password

1. Open the File Security Setup view.
2. Click the **Change Password** button. The Enter Master Password view appears.



3. Enter the password, then press **ENTER**. The New Password view appears.



 Accept Password icon



Cancel icon

4. Enter a new password in both the **Password** and **Confirm** fields.
5. Click the **Accept Password** icon.
6. Click the **Cancel** icon to exit the function.

Clearing the Master Password

1. Click the **Clear Password** button. The Enter Master Password view appears.



Press the **ESC** key at any time to cancel deleting the password.

2. Enter the password, then press **ENTER**.

User Password

After a master password is entered, a user password can also be entered. This is optional, but useful when more than one user is working with the case and you do not want to give access to the security setup.

The following restrictions are applied to the user password:

- Cannot be the same as the master password.
- Cannot be less than six characters or more than 24 characters.

If a case is opened with a user password, then the File Security Setup view is not available.

Adding a User Password

1. Enter the password in the **Password** field.
2. Retype the password in the **Confirm** field to confirm the password.
3. Click the **Accept user password** icon.

After accepting the user password, the **Accept user password** icon is greyed out and the two icons to the left of it become active.



Accept user password icon

Viewing the User Password

1. Click the **View user password** icon. The Enter Master Password



View user password icon

view appears.

Figure 11.67



2. Type the correct password.
3. Press **ENTER** and the user password can be viewed.
4. Click the **View User Password** icon again to hide the user password.

Clearing the User Password



Clear user password icon

1. Click the **Clear user password** icon. The Enter Master Password view appears.

Figure 11.68



Press the **ESC** key at any time to cancel this operation.

2. Type the correct password.
3. Press **ENTER**.

Setting a Time Restriction

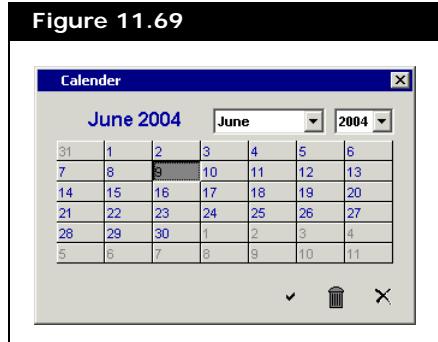
To place a time restriction on a user password:

1. Click the **Open Calendar View** icon. The Calendar view



Open Calendar View icon

appears.



2. From the month drop-down list, select the month that the password expires.
3. From the year drop-down list, select the year that the password expires.
4. From the calendar, select the day that the password expires. This date is the time restriction date.
5. Click the **Accept and close** icon.

 Accept and close icon

Refer to the table for the functionality of the remaining icons.

Name	Icon	Description
Clear good until date		Clears the selected time restriction date.
Cancel		Cancels the action.

Hardware Locks

Cases can also be locked to hardware devices on your machine or a client machine for additional security. The **Hardware Locks** button scans the current machine for a possible lock, while the **Advanced** button lets you enter information that is found on a client's machine.

Scanning for Lock Codes

1. Click the **Hardware Locks** button. The Scan All Locking Codes view

appears.

Figure 11.70

	Lock Code	Accept
ID Prom	Not Found	<input type="checkbox"/>
IP Address	2b722	<input type="checkbox"/>
Disc ID	2bb23	<input type="checkbox"/>
Host Name	27049	<input type="checkbox"/>
Ethernet Address	30303	<input type="checkbox"/>
Net IPX Address	Not Found	<input checked="" type="checkbox"/>
Net Serial Number	Not Found	<input checked="" type="checkbox"/>
Computer ID	2985d	<input type="checkbox"/>
Green Key	c2c5	<input type="checkbox"/>
CPU Serial Number	Not Found	<input checked="" type="checkbox"/>

2. Click the **Scan** button to scan your system for all the locks listed. The Lock code column either displays the lock code for the associated hardware device or “Not Found” if no lock code is available.
3. Check the **Accept** checkbox for each of the lock codes you want to lock the case to.
4. Click the **Exit View** icon to accept the changes. The selected lock codes appear in the Hardware Locks table.

You can click the **Cancel** icon to close the Scan All Locking Codes view without accepting any changes.



Exit View icon



Cancel icon

Specifying Lock Codes

To enter a specific lock code.

1. Click the **Advance** button. The Advanced Lock Selection view appears.

Advanced Lock Selection	
Lock Type	Network Serial Number <input checked="" type="checkbox"/>
Lock Code	<input type="text"/> <input type="button" value="X"/>

2. From the Lock Type drop-down list, select the hardware device being locked to the case.
3. In the **Lock Code** field, specify the lock code that corresponds to the selected lock type.
4. Click the **Accept** icon and the lock code displays in the Hardware Locks table.

You can click the **Cancel** icon to close the view without accepting any changes.



Accept icon



Cancel icon

You can use the Echo ID tool to help you determine the lock codes for your computer. Refer to [Section 11.16 - Echo ID](#) for more information.

Removing Hardware Locks

To remove a single hardware lock:

1. Select the hardware lock you want to remove from the Hardware Locks table.
2. Click the **Remove** button. The Enter Master Password view appears.

Figure 11.72



Press the **ESC** key at any time to cancel this operation.

3. Type the correct password and press **ENTER**.

To remove all hardware locks:

1. Click the **Remove All** button. The Enter Master Password view appears.

Figure 11.73



Press the **ESC** key at any time to cancel this operation.

2. Type the correct password.
3. Press **ENTER**.

11.15.4 Unlocking a Case

Although a case is locked and all passwords are entered, the case can still be unlocked without clearing all the passwords and locks.

1. Click the **Case Security is Currently Active** icon. The Enter



Case Security is
Currently Active icon

Master Password view appears.



2. Type the correct password.

Press the **ESC** key at any time to cancel this operation.

3. Press **ENTER**. The **Case Security is Currently Active** icon changes to **Case Security is Currently Disable** icon which indicates that the case is unlocked.



Case Security is
Currently Disable icon

To lock the case again, click the **Case Security is Currently Disable** icon and the file security is automatically activated.

11.15.5 Runtime Mode

A Runtime case is a version of an application that has a limited set of capabilities compared to the standard version of that application.

The Runtime tab allows you to set the restrictions for a runtime case.

Runtime licensing is a special and restricted type of license that Honeywell grants users. A user that has a runtime license is permitted to load any UniSim Design runtime case or may be restricted to only load runtime cases with a certain Runtime ID or from a certain author (see below). In all situations a user with a runtime license can not add or delete unit operations and hence can not build new cases or modify the flowsheet topology of existing cases. A runtime license user can not load non-runtime cases. The runtime license is authenticated by the UniSim Design security system and the license file that is provided by Honeywell.

In the use of runtime licensing, there is a runtime case originator or author and there is a runtime case user. This capability is very useful for process licensors, engineering firms or other solution providers. In fact, one company or department can use a network license file with the same RunTime-Author entry. This way when they publish runtime cases, they all are bound against the same author and the users of these cases only need a runtime license permitting use of this author or supplier's cases.

A UniSim Design runtime case is typically created by the original developer of the model. As the creator of the runtime case, you can create it in one of two ways:

1. If you have RunTime-Author capability in your license file, you would also have present in this license file a certain author id. In

this situation you can create any number of different runtime cases, all marked with the same author (or runtime id). Then any of your clients or affiliates which have a runtime license authorizing the running of your authored cases can utilize these cases. Contact Honeywell if you would like to author runtime cases in this manner. Honeywell will supply you with a RunTime-Author license.

2. If your license file does not have RunTime-Author capability, or if you do not wish to convert a case to runtime as this author, you can also use a unique RunTime ID (RTID). This RTID is automatically generated for each case each time you convert the case to runtime. In this situation, you will need to contact your client or affiliate and inform them that they need to request a runtime license file from Honeywell which permits the use of runtime cases with this RTID.

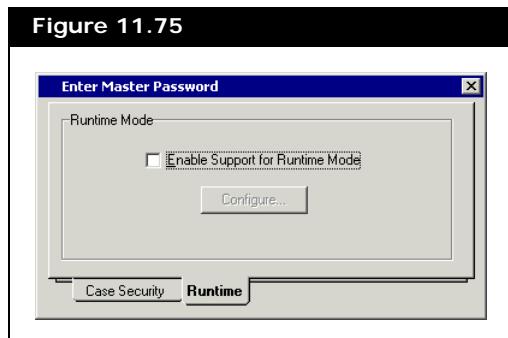
Depending on the original contract terms with Honeywell, a runtime licensed user may also obtain an open runtime license which means that they can open and use any runtime cases with any RTID but not if that RTID is the same as the author (bound to author).

There is also some capability for the creator of the runtime case to permit or restrict what unit operations or specifications may be changed. See below for more details on this.

Activate Runtime Mode

Use this procedure to activate the runtime mode in UniSim Design:

1. In the menu bar, select **Tools | Case Security** command.
The Enter Master Password view appears.
2. Click on the **Runtime** tab.



3. Select the **Enable Support for Runtime Mode** checkbox.
4. Click the **Configure** button. The Runtime Mode Configuration view appears.
5. In the **General** tab, select the appropriate checkbox to activate the options you want.

6. In the **Specifications** tab, select the specifications you want to enable for other people to modify.
7. In the **Access** tab, select which objects are accessible by other people.

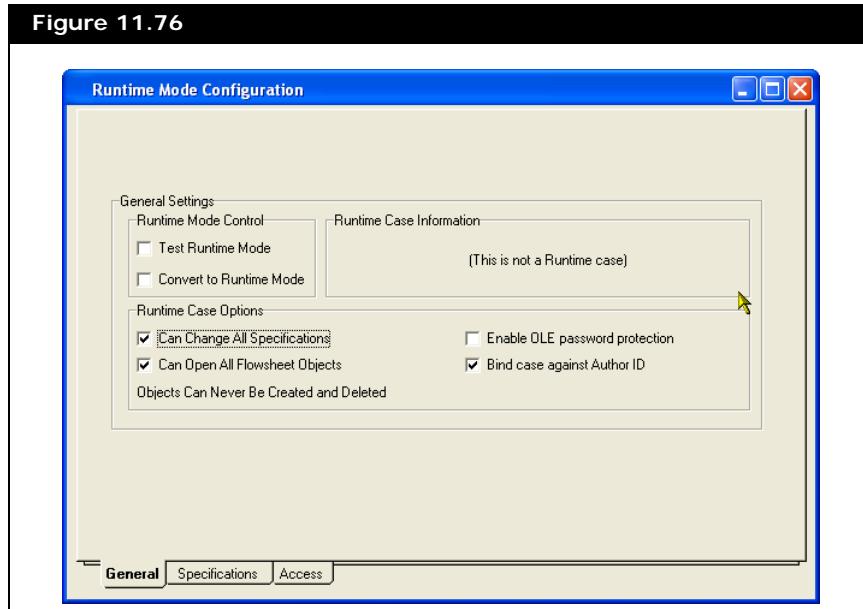
Runtime Mode Configuration View

The Runtime Mode Configuration view allows you to create a runtime case with a custom set of restrictions. The following tabs are available in the Runtime Mode Configuration view:

- General
- Specifications
- Access

General Tab

The General tab allows you to set general restrictions in a runtime case.



The General tab contains the following groups:

- Runtime Model Control
- Runtime Case Options
- Runtime Case Information

Runtime Mode Control Group

The Runtime Mode Control group contains the following checkboxes:

Checkbox	Descriptions
Test Runtime Mode	You can check this checkbox to test the current case in runtime mode under the settings specified in the Runtime Case Options and Specifications tab.
Convert to Runtime Mode	You can check this checkbox to convert the current case to a runtime case under the settings specified in the Runtime Case Options group and Specifications tab.

Once the case is converted into runtime mode, you have to exit UniSim Design to restore the standard operation mode.

Runtime Case Options Group

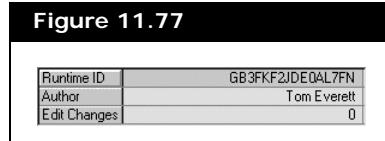
The Runtime Case Options group contains the following checkboxes:

Checkbox	Descriptions
Can Change All Specifications	Allows you to change all the specified properties (blue values) of the unit operations in the runtime case.
Enable OLE password protection	<p>Enables you to apply the OLE password protection feature.</p> <p>When this protection feature is activated, only OLE access to the case is allowed if the runtime password is supplied. The OLE methods available are:</p> <ul style="list-style-type: none"> • OpenLockedOLEWithPassword • put_VisibleWithPassword • PlayScriptWithPassword • PlayScriptRelativeToWithPassword <p>So, if a case is OLE locked then,</p> <ul style="list-style-type: none"> • OpenWithPassword is replaced by OpenLockedOLEWithPassword • PlayScript is replaced by PlayScriptWithPassword • PlayScriptRelativeToWithPassword • put_Visible is replaced by put_VisibleWithPassword
Can Open All Flowsheet Objects	<p>Enables you to toggle between access to or restriction from all flowsheet objects when in runtime mode.</p> <p>This feature overrides any protections specified on the Access tab.</p>
Bind Case Against Author ID	Enables conversion to runtime case with the Runtime ID being set equal to the author. This allows the user of the runtime case to use an existing license file which permits the loading of this author's cases. the user can use multiple cases from this author or publishing company.

Runtime Case Information Group

The Runtime Case Information displays the Runtime ID, Author, and the number of edit changes in a runtime case. The Runtime Case Information table appears only when the case is confirmed to run in runtime mode. The Author entry may be blank if their license file does

not have a runtime Authoring entry. One can still convert a case to runtime but a unique RTID will be generated.



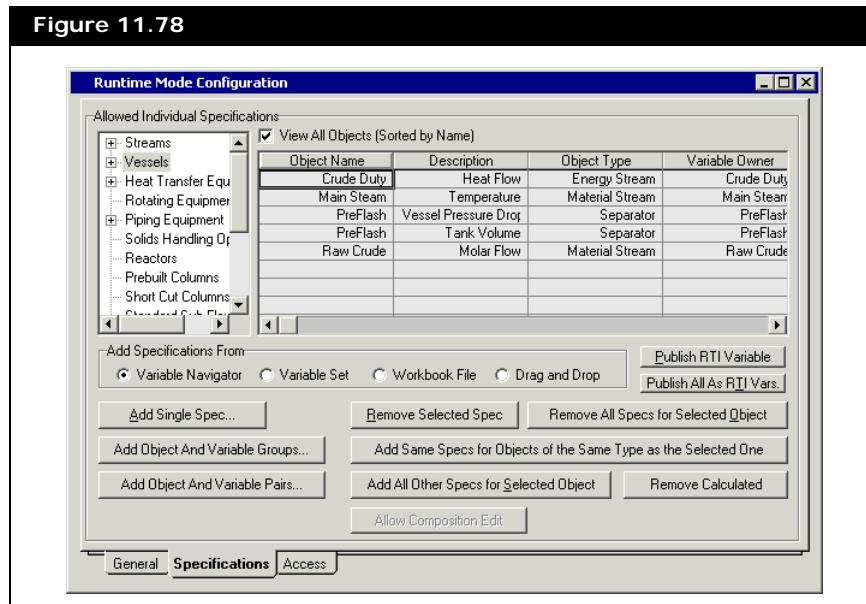
Each time a runtime case is created, a unique Runtime ID (RTID) is assigned to the case as long as the Bind case against Author ID option is not selected. The Author of the runtime case has to advise Honeywell of the RTID so that a license can be issued to the users to run the case.

A runtime case can only be opened in standard mode by the original Author. When a runtime case is saved, it will store the name of the Author in the case. If the name of the user does not match the runtime author license, then the case opens in runtime mode and the user can only use the runtime case under the restrictions pre-set by the original

Specifications Tab

The Specifications tab will appear blank if the **Can Change All Specifications** checkbox is selected.

The Specifications tab allows the original Author of the runtime case to select the parameters that can be modified in runtime mode.



Select the method for importing the changeable parameters by selecting one of the four radio buttons available in the Add Specification From group.

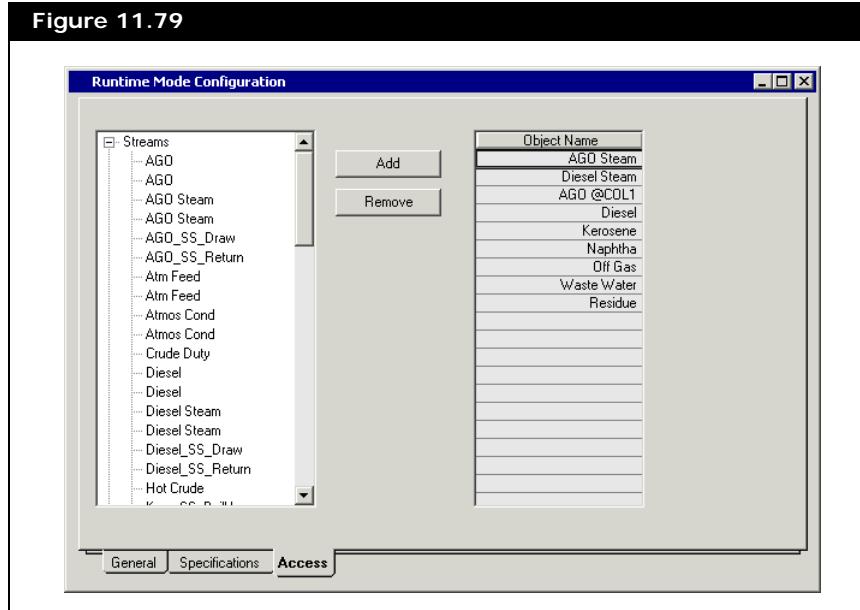
The parameters you select appear in the table on the Specifications tab, and you can view the Object Name, Description, Object Type, and Variable Owner for each parameter. The parameters you can modify appear in blue text when you run the runtime case.

You can modify only the parameters selected on the Specifications tab when the case is in runtime mode. All other parameters are view-only (black text) for the runtime case.

Access Tab

The Access tab will appear blank if the **Can Open All Flowsheet Objects** checkbox is selected.

The Access tab enables you to select which objects in the simulation case can be viewed by other people.



To make an object available for modification to other users:

1. In the object tree, click the + icon to expand the branches and select the object you want to add.
2. Click the **Add** button.

To remove an object:

1. In the **Object Name** list, select the object you want to remove.
2. Click the **Remove** button.

11.16 Echo ID

The Echo ID tool allows you to scan your computer and display all of the

available locking codes. To open the Scan All Locking Codes view, click the Echo ID command in the Tools menu.

Figure 11.80

Scan All Locking Codes	
	Lock Code
ID Prom	Not Found
IP Address	2b722
Disc ID	2bb23
Host Name	27049
Ethernet Address	30303
Net IPX Address	Not Found
Net Serial Number	Not Found
Computer ID	2985d
Green Key	c2c5
CPU Serial Number	Not Found

11.17 Correlation Manager

Refer to [Chapter 3 - Streams](#) in the [UniSim Design Operations Guide](#) for more information about managing property correlations in individual streams.

The Correlation Manager allows you to manage individual property correlations or property correlation sets that can be applied to all streams in your case.

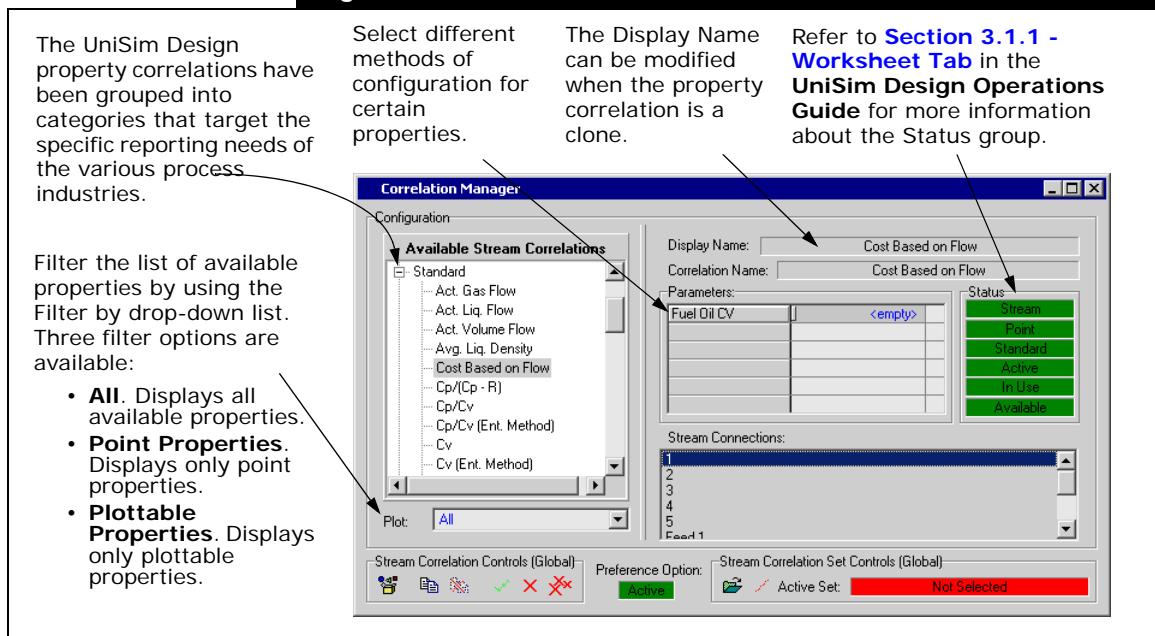
UniSim Design has two kinds of settings for property correlations. A property correlation or correlation set that is added to a stream from the stream property view has a local setting. A property correlation or correlation set that is added to a stream from the Correlation Manager view has a global setting.

The property correlation sets can be defined and saved external to the case and can be read into any other simulation case.

To access the Correlation Manager, select the Correlation Manager

command from the Tools menu.

Figure 11.81



The Correlation Manager view contains three groups and a Preference Option status bar. The Configuration group contains a list of property correlation types. The Stream Correlation Controls (Global) group contains the following icons:

Name	Icon	Description
Scan System Correlations		Scan all system correlation files. Click this icon to manually scan the system registry and build a list of available property correlations.
Clone Selected Correlation		Click to clone a selected property correlation. You can only clone property correlations with variable parameters.
Delete Cloned Correlation From List		Click to delete the selected property correlation clone. You can only delete property correlation clones.
Activate Selected Global Correlation		Click to add the selected global property correlation to all the streams.
Remove Selected Global Correlation		Click to remove the selected global property correlation from all the streams.
Remove All Global Correlation		Click to remove all the global property correlation from all the streams.

The Preference Option status bar indicates whether the Activate

Property Correlations checkbox on the Options page, Simulation tab of the Session Preferences view is checked or not.

The stream property view lets you create your own group of property correlations called a correlation set. The Stream Correlation Set Controls (Global) group lets you to select a correlation set and globally apply it to all streams in the case.

You can only create a correlation set in the stream property view.

Name	Icon	Description
View Global Correlation Set List		Opens the Correlation Set Picker. From this view, select the correlation set you want to apply to all case streams. The Correlation Set Picker will display "File has not been created" until you have saved your first set.
Remove Global Correlation Set		Click to remove the global correlation set displayed in the status bar from all the streams. The Correlation Manager will only remove the correlation properties that are part of the selected global set. Any other correlation properties that have been globally (or locally) added to the streams are not removed.
Active Set status bar		Provides description on the status of the active selected global correlation set. <ul style="list-style-type: none"> • Red status bar. Indicates that no global correlation set is selected for the streams. • Green status bar. Indicates that a global correlation set is selected for the streams. • Yellow status bar. Indicates that the global correlation set has been modified. This is when correlation properties that are not part of the set have been globally added to all streams (activated), or when properties of this set have been globally removed from all streams (deactivated).

11.17.1 Adding Property Correlations to Streams

Use this procedure to add a global property correlation to the streams.

1. From the list of Available Correlations, click the + icon to expand the tree, displaying the available property correlations for each type. All property information for the selected property correlation appears on the table to the right.
2. Certain properties require qualifiers before they are fully configured. Click the appropriate field and specify any values that are required.
3. Click the **Activate Selected Global Correlation** icon in the Stream Correlations Controls (Global) group. The Global Active status bar changes from red to green.



Activate Selected Global Correlation icon

11.17.2 Removing Property Correlations from Streams

Use this procedure to remove global property correlations from the streams.

1. From the Available Correlations list, select the property correlation you want to remove from each stream in your case. Click the + icon to expand the tree, displaying the available property correlations for each type. The property information for the selected property correlation appears on the table to the right.
2. Click the **Remove Selected Global Correlation** icon in the Stream Correlations Controls (Global) group. The Global Active status bar changes from green to red.
3. To remove ALL global property correlations and active global correlation sets from the streams, click the **Remove All Global Correlations** icon in the Stream Correlations Controls (Global) group.



Remove Selected Global Correlation icon



Remove All Global Correlations icon

11.17.3 Cloning Property Correlations

Any property correlation that contains changeable parameters can be cloned. When a correlation is globally activated, all streams in the case use the same calculation specified by that correlation. When you change a correlation's parameter, all the streams in the case use the new parameter to calculate that correlation's value.

If you require a correlation to have different parameter values for different streams, you need to create a clone for that correlation. When you create a clone, you can specify a unique parameter value for that correlation and then add the clone to the appropriate streams.

Do the following to create a clone of a property correlation:

1. From the Available Correlations list, select the property correlation you want to clone. Click the + icon to expand the tree, displaying the available property correlations for each type.
2. Click the **Clone Selected Correlation** icon in the Stream Correlations Controls (Global) group. The new property correlation is added to the Available Correlations tree under the Clone branch. UniSim Design automatically names each cloned property correlation using the original property correlation name and "**_Clone-1**". The number at the end increases with each clone of the same property correlation.



Clone Selected Correlation icon

3. Rename the clone property correlation in the **Display Name** field if required. You can only rename cloned property correlations and each clone name must be unique.

11.17.4 Deleting Cloned Property Correlations

1. From the Available Correlations list under the Clone branch, select the clone property correlation you want to delete. Click the + icon to expand the Clone branch displaying the available clone property correlations.
You can only delete clone property correlations that are not used by any streams.
2. Click the **Delete Cloned Correlation From List** icon in the Stream Correlations Controls (Global) group. The selected clone property correlation is removed from the Available Correlations tree under the Clone branch. If the selected clone is being used by any stream within the case, it cannot be deleted.

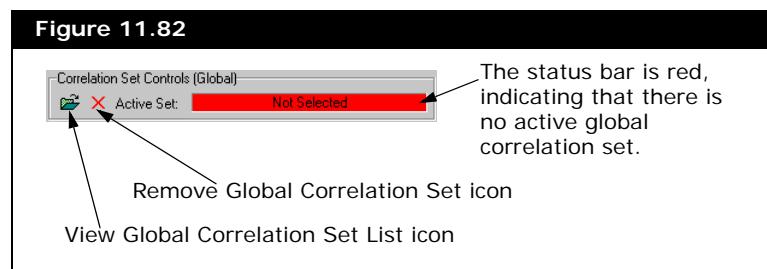


Delete Cloned Correlation
From List icon

11.17.5 Adding Correlation Sets to Streams

Use this procedure to add a global correlation set to all the streams in the case.

1. Before selecting a correlation set, there must be no active global correlation set.



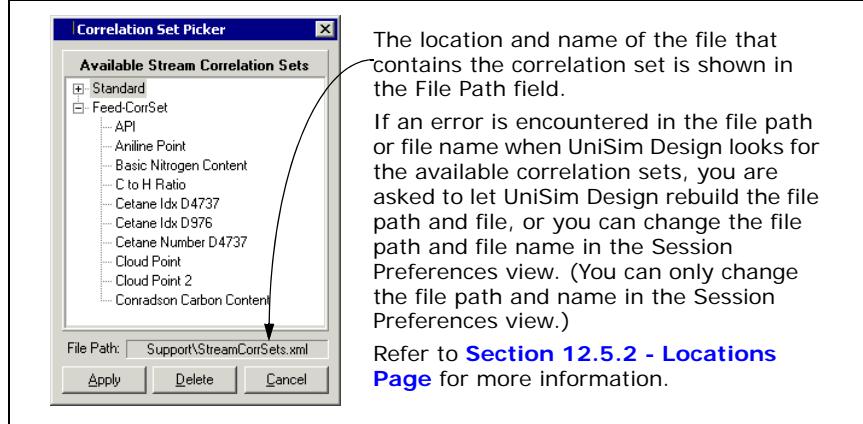


View Global Correlation Set List icon

The Correlation Manager will not display the Correlation Set Picker view if a set is already globally active. Only one correlation set can be globally active at a time.

- Click the **View Global Correlation Set List** icon. The Correlation Set Picker view appears.

Figure 11.83



If you have a new product installed the Support\StreamCorrSets.xml file will not exist until you actually save a set.

UniSim Design creates the file the first time you want to save a set. The program manages the file after this, but should only have to create the file once. If you are looking in the Support directory the file will not exist there until the user has saved a correlation set using one of the stream properties views (Refer to [Section 3.1.1 - Worksheet Tab](#) in the UniSim Design Operations Guide).

- Select the correlation set you want from the view. You can click the + icon to see what property correlations the correlation set contains.
- Click the **Apply** button. The selected correlation set is now the active global correlation set. All the streams in the case contain the selected correlation set and any new stream added to the case will automatically contain the selected correlation set.

Whenever a correlation/correlation set is applied to the streams, a check is made of the correlation type against the fluid type of each stream. If the Correlation Manager encounters a problem, it will send a warning to the trace window.

The Active Set status bar turns green and displays the selected global

correlation set.



11.17.6 Deleting a Correlation Set

Refer to [Deleting a Correlation Set](#) in [Section 3.1.1 - Worksheet Tab](#) from the **UniSim Design Operations Guide** for more information.

11.17.7 Gas Properties Correlation

The Gas Properties in the Correlation Manager consists of the following seven correlations:

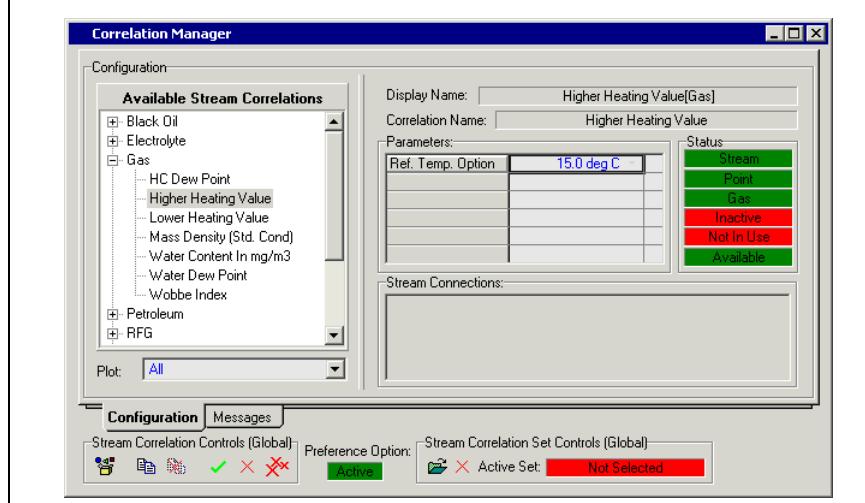
- HC Dew Point
- Higher Heating Value
- Lower Heating Value
- Mass Density (Std. Cond)
- Water Content In Mg/m³
- Water Dew Point
- Wobbe Index

Refer to [Adding a Property Correlation](#) in [Section 3.1.1 - Worksheet Tab](#) from the **UniSim Design Operations Guide** for more information on the Correlation Picker.

These correlations are grouped within the Gas correlation type. The Gas type is shown on the Available Correlations window of the Correlation Picker and Correlation Manager views. You can access the seven Gas correlations by double-clicking on the Gas correlation type to expand

the list.

Figure 11.85



The following four Gas correlation types allow you to specify the calculation's reference temperature:

- Higher Heating Value
- Lower Heating Value
- Mass Density (Std. Cond)
- Wobbe Index

For more information on the Correlation Manager, refer to [Section 11.17 - Correlation Manager](#).

Properties

The Gas correlation uses the methods and data from ISO 6976:1995(E). If the stream contains components that are not supported by this data, then the data for the hydrocarbon with the nearest molecular weight Alkane is used. If the molecular weight is greater than decane (C10), then the data for decane is assumed. ISO data has been provided to support the following components:

Components				
Methane	n-Decane	n-C21	Propene	M-Cyclohexane
Ethane	n-C11	n-C22	Methanol	CO
Propane	n-C12	n-C23	EGlycol	CO2
i-Butane	n-C13	n-C24	TEGlycol	H2S
n-Butane	n-C14	n-C25	Benzene	Ammonia
i-Pentane	n-C15	n-C26	Toluene	H2O
n-Pentane	n-C16	n-C27	E-Benzene	Hydrogen

Components				
n-Hexane	n-C17	n-C28	124-M-Benzene	Nitrogen
n-Heptane	n-C18	n-C29	Cyclopentane	Argon
n-Octane	n-C19	n-C30	Cyclohexane	Oxygen
n-Nonane	n-C20	Ethylene	M-Cyclopentane	

Higher Heating Value

The higher heating value is the amount of heat obtained during combustion when the water produced in the combustion is condensed (when the water is in liquid form).

$$HHV = \frac{\sum_i x_i \times HHV_i}{Z} \quad (11.1)$$

where:

HHV = Overall higher heating value

HHV_i = Higher heating value of component i

Lower Heating Value

The lower heating value is the amount of heat obtained when this water is not condensed (when the water is in vapour form).

$$LHV = \frac{\sum_i x_i \times LHV_i}{Z} \quad (11.2)$$

where:

LHV = Overall lower heating value

LHV_i = Lower heating value of component i

Hydrocarbon & Water Dew Point

The dew point is the temperature at a given pressure at which the first drop of liquid starts to form. Hydrocarbon and water dew points

indicate the type of liquid formed.

In UniSim Design, the dew point is calculated by using a flash to calculate the temperature at which the vapour fraction is equal to one. When the vapour fraction is equal to one, the vapour is said to be saturated or at steam pressure. If the case does not contain water, the temperature at which the vapour fraction equal to one is reported as the hydrocarbon dew point. If the case has water as one of the components, the type of liquid phase formed is reported with its corresponding dew point temperature. Then the temperature is decreased until a second liquid phase forms, the temperature at which this occurs is reported as the other dew point.

Mass Density at Standard Conditions

The mass density is reported at 1 atm and 15°C (288.15K). From the Ideal Gas law:

$$PV = nZRT \quad (11.3)$$

where:

P = Pressure (kPa)

V = Volume

n = Number of moles

Z = Compressibility factor

R = Gas constant (8.3145 J/mol K)

T = Temperature (K)

The molar density can be expressed as:

$$\frac{1}{V} = \frac{P}{ZRT} \quad (11.4)$$

The mass density is therefore:

$$\rho = \frac{\left(\sum_i x_i \times MW_i \right) \times P}{Z \times R \times T} \quad (11.5)$$

where:

ρ = Mass density (kg/m^3)

x_i = Mole fraction of component i

MW_i = Molecular weight of component i

Water Content

The water content is the mass of water per unit volume of each phase. It is expressed as mg/m^3 .

Wobbe Index

The Wobbe Index (or Wobbe Number) is a measure of how much heat is released when gas is burnt.

The Wobbe Index is calculated by dividing the HHV (higher heating value) by the square root of the density relative to air.

$$Wobbe = \frac{HHV}{\sqrt{\rho_{relative}}} \quad (11.6)$$

The higher heating value is the amount of heat obtained during combustion when the water produced in the combustion is condensed.

Density Relative to Air

The density relative to air is calculated by dividing the equation for the density of the stream by the equivalent one for the density of air at 1 atm and 15°C.

$$\rho_{relative} = \frac{\sum_i x_i \times MW_i}{Z} \times \frac{Z_{air}}{MW_{air}} = \frac{\sum_i x_i \times MW_i}{Z} \times \frac{0.99958}{28.962} \quad (11.7)$$

where:

ρ = Density relative to air

$$MW_{air} = \text{Molecular weight of air}$$

11.17.8 RVP Properties

Refer to
<http://www.api.org/>
<http://www.astm.org/>
 for more information.

The RVP properties uses the methods and data constructed by the company API. The RVP properties consist of the following eight correlations:

- Reid VP at 37.8 C
- True VP at 37.8 C
- API 5B1.1
- API 5B1.2
- ASTM D323-73/79
- ASTM D323-82
- ASTM D4953-91
- ASTM D5191-91

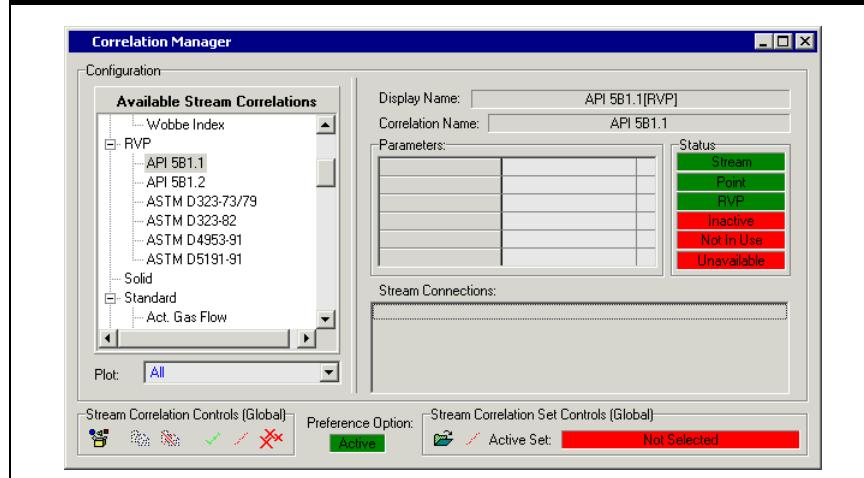
The Reid VP at 37.8 C and True VP at 37.8 C correlations are grouped within the Standard correlation type, and the other six correlations are grouped under the RVP correlation type.

Refer to the section on **Adding a Property Correlation** in Section 3.1.1 - Worksheet Tab from the **UniSim Design Operations Guide** for more information.

The RVP and Standard types are shown on the Available Correlations window of the Correlation Picker and the Correlation Manager views. (For more information on using the Correlation Manager, refer to **Section 11.17 - Correlation Manager**.)

You can access the eight correlations by double-clicking their corresponding correlation type to expand the list.

Figure 11.86



Properties

The following is a brief description of each of the correlations listed under the RVP type.

API 5B1.1 (Naphtha)

This property correlation is useful for gasoline and finished petroleum products, but not crude or oxygenated blends. The TVP is correlated against the RVP, temperature, and slope of the ASTM D86 distillation curve at the 10% point. This property solves the corrected version of the API databook equation of the correlation for the RVP. A recognized limitation of the API Naphtha correlation is that the D86 10% point can have a similar gradient for vastly different streams.

API 5B1.2 (Crude)

This property correlation is generally used for condensate and crude oil systems (typically wide boiling preprocessed hydrocarbons). TVP is correlated against RVP and the temperature. This property solves the API databook equation of the correlation for RVP.

The correlation is based on data from 1959, but it is popular with engineers for its quick and dirty calculations.

ASTM D323-73/79

This correlation is also known as P323. The pressure is adjusted at the RVP reference temperature until the vapour to liquid ratio is 4:1 by volume. This correlation is essentially the same as the Reid VP at 37.8 C correlation, except it is not on a dry basis and the flash method used is the same for the rest of the flowsheet.

ASTM D323-82

This is the standard and accepted procedure for RVP lab measurement. Liquid hydrocarbon is saturated with air at 33°F and 1 atm pressure. Since the lab procedure does not specify that the test chamber is dry, the air used to saturate the hydrocarbon is assumed to be saturated with water.

This air-saturated hydrocarbon is then mixed with dry air in a 4:1

volume ratio and flashed at the RVP reference temperature, such that the total volume is constant (since the experimental procedure uses a sealed bomb). The gauge pressure of the resulting mixture is then reported as the RVP.

ASTM D4953-91

This correlation is for oxygenated gasoline. It is the same as the D323-82 test method, except everything is on a completely dry basis (i.e., the air is not saturated with water).

ASTM D5191-91

This was developed for gasoline and gasoline-oxygenate blends as an alternative to the D4953-91 test method. In the experimental procedure, the hydrocarbon is saturated with dry air and then placed in an evacuated bomb with five times its volume. The total pressure is then converted to a dry vapour pressure equivalent (DVPE) and reported as the RVP.

The method used is to mix near vacuum air at 0.01 psia and 100°F with hydrocarbon at 1 atm and 33°F in the ratio 4:1. This is then flashed at constant volume at the RVP reference temperature. The pressure is then converted to a DVPE and reported as the RVP.

Conditions for Using RVP

In order to apply some of the RVP correlations to your stream, the components in the stream need to comply with the correlation.

Nitrogen, oxygen, and water need to be present in the stream if you are using the following RVPs:

- ASTM D323-82
- ASTM D4953-91
- ASTM D5191-91

If all three components are not present within your stream and you want to use any of the above three RVPs, UniSim Design will administer a warning in the trace window.

Electrolyte Components

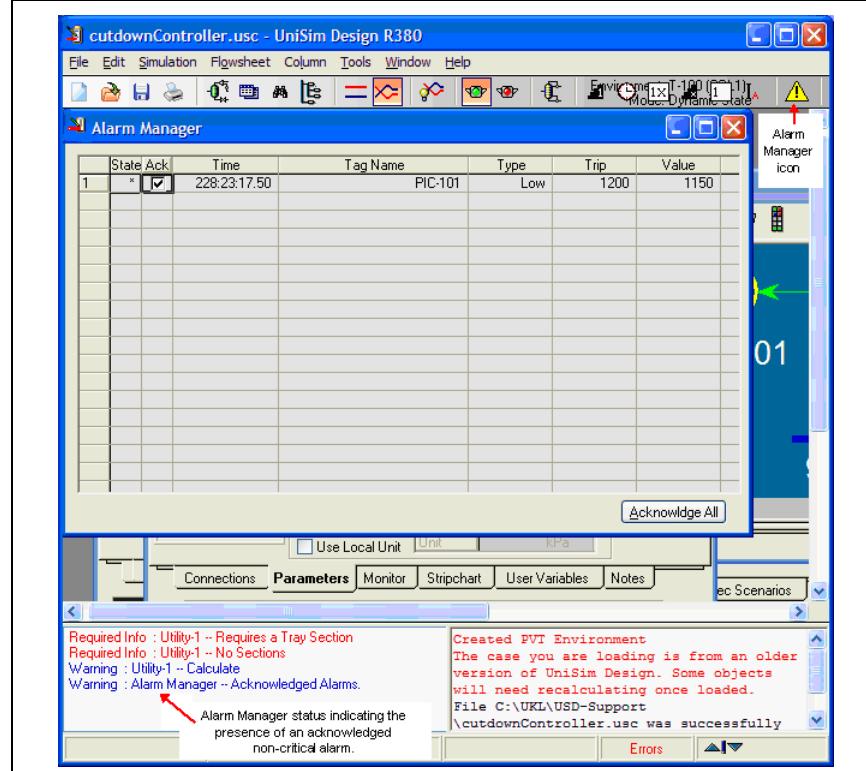
Electrolytes are the only type of components which cannot be used with

any of the RVP correlations. If an Electrolyte component is present in the stream and you try to add a RVP correlation, UniSim Design will not allow you to use it. If the RVP correlation is already applied to a stream and you add the Electrolyte component afterwards, UniSim Design will automatically remove the electrolyte component from that given stream.

11.18 Alarm Manager

The Alarm Manager is used to view and acknowledge the current status of raised alarm events by suitably-configured equipment items. Currently Controller P V alarms are supported. The Alarm Manager is available only in dynamic mode and can be quickly accessed in the main and column environments via the Dynamics toolbar icon .

Figure 11.87



11.18.1 Opening the Alarm Manager

There are a number of ways to open the Alarm Manager when running

a dynamic case:

- Click the **Alarm Manager** icon in the toolbar (A yellow warning symbol)
- Click the **Alarm Manager** menu entry in the Tools menu
- Double-click on an Alarm Manager message in the status information area when the Alarm Manager status appears as a result of an alarm event.

11.18.2 Understanding the Alarm Manager

The Alarm Manager displays current alarm events in a strict chronological (simulation time) order, with the most recently occurring alarm appearing at the top of the list.

A vertical scrollbar automatically appears to accommodate large lists of alarms greater than the default window size.

For each alarm event an entry is created in the alarm display manager displaying the following information:

Event	Description
State	Whether the object is actually in alarm or not. An asterisk in this column indicates that this item is still in alarm. (This allows the view to differentiate between items in alarm and items which have moved from an in-alarm state to a no-alarm state without the user having acknowledged the occurrence of the alarm).
Ack	The checkbox indicates whether the alarm has been acknowledged by the user. Clicking on the checkbox when it is an unchecked state acknowledges the alarm. Clicking on an already-checked field has no effect.
Tag Name	The name of the equipment item. Depending on context, the displayed name will be the plain equipment name - or if the equipment item is present in a sub-flow sheet - the fully qualified equipment name. Clicking on this display pops-up the view of the equipment item in question.
Type	Describes the Alarm type - may be one of HighHigh, High, Low or LowLow.
Trip	Displays the trip point of the breached alarm threshold.
Value	Displays the current value of the displayed variable.

Alarming is currently supported for the PV of controller. Refer to [Alarms Page](#) from the **UniSim Design Operations Guide** for more information.

Alarm events from suitably configured equipment objects (i.e. appropriate alarm limits have been set) appear on the alarm display of an executing dynamic case under the following circumstances:

- The PV of a monitored equipment object breaches an alarm limit.
- An alarm limit is changed so that the current value of the item now breaches the modified limit.

Alarm events are automatically removed from the alarm display of an executing dynamic case under the following circumstances:

- The PV of an **acknowledged** alarm event leaves an alarm limit.
- An alarm limit is changes so that the PV of an **acknowledged** event no longer breaches that limit.

An **unacknowledged** alarm is not automatically removed from the display under the above two conditions. Instead, only the asterisk declaring that it is in alarm (in the State column of the display) is removed. The alarm remains until the user explicitly acknowledges that alarm (or clears it as part of a Clear All operation via the Acknowledge pushbutton in the display). Once acknowledged the alarm event is immediately removed from the display.

Multiple Alarm Events for the Same Point

A single point may appear in the alarm list more than once if multiple alarm limits have been breached. For example an uncorrected **Low** alarm may transition to a **LowLow** - in this case separate entries for both the **Low** and **LowLow** conditions are created. As described above, neither of these entries would be removed from the alarm list if the point returns to a healthy condition unless the alarm has been explicitly acknowledged.

In the above example, a point value returning from a level triggering a **LowLow** alarm condition to a healthy value would pass through a number of states - depending on whether both, one or neither of the alarm conditions had been acknowledged:

LowLow and Low unacknowledged:	Both remain in the alarm manager as unacknowledged entries.
LowLow acknowledged:	LowLow removed from list once LowLow limit (plus tolerance) surpassed. Low entry remains.
Low and LowLow acknowledged	Each removed in turn as respective limits (plus tolerance) surpassed.

Similar rules apply to alarms breaching **High** and **HighHigh** limits - or in the case of a wildly transitioning point - between the **Low** and **High** limit sets.

As described in the section above, acknowledging an alarm condition when that condition is no longer active results in its immediate removal from the list.

Acknowledging an Alarm

To acknowledge an individual alarm:

- Click the checkbox in the **Ack** column

To acknowledge all alarms on the display:

- Click the **Acknowledge All** pushbutton located at the bottom right of the Alarm Manager Display

Status Reports

The user is alerted to alarm-related activity by entries in the Status Window.

Messages vary according to the severity of the current alarm manager contents in the following order of precedence:

1. If an **unacknowledged** high-importance alarm is present (e.g. **HighHigh** or **LowLow**) this information is presented as a high-priority (red) message.
2. If an **unacknowledged** medium importance (e.g. **High** or **Low**) this information is presented as a high-priority (red) message.
3. If an **acknowledged** high importance event is present this is reported as a warning (blue) message.
4. If an **acknowledged** medium-importance event is present this is reported as a warning (blue) message.

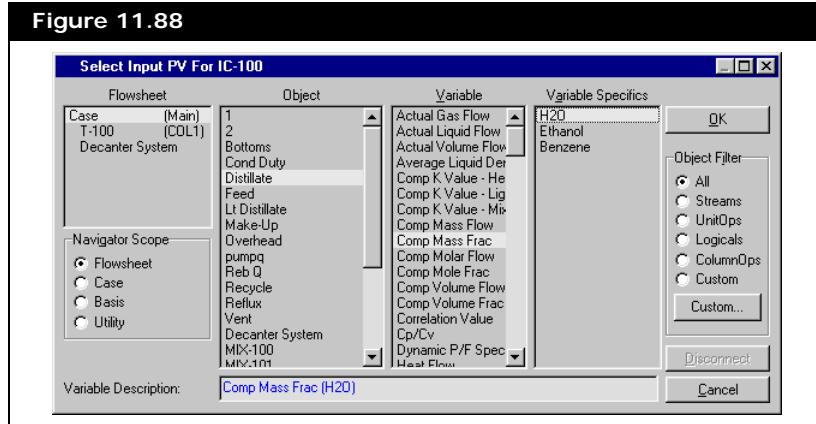
The Alarm Manager can be displayed by double-clicking on any of those alarms.

11.19 Variable Navigator

The Variable Navigator is one of the two navigational aids that can be used to bring the multi-flowsheet architecture into a flat space. It is also used when browsing for variables, such as selecting a process

variable for a controller or a strip chart.

Figure 11.88



The Variable Navigator uses a similar structure to the Object Navigator, but allows for a more detailed search.

11.19.1 Using the Variable Navigator

Refer to [Section 7.17 - Object Navigator](#) for details on list filtering.

When selecting a variable, work through the groups from left to right. The Object Filter eliminates selections from the Object group.

You can click the **Cancel** button at any time to close the Variable Navigator without accepting any changes.

Selecting a Process Variable

1. Select one of the radio buttons in the Navigator Scope group. The list located above this group changes depending on the radio button selected.
2. From the list, select either the flowsheet, case, basis, or utility in which the process variable is located.
3. From the list of available objects, select the object with the variable you want to use.
4. From the list of available variables, select the variable you want to use.
5. Certain variables (such as component specific variables), require further specification. From the list of variable specifics, select the qualifier for the variable.
6. Enter a more detailed description of the variable in the **Variable Description** field or leave the default description.

7. Click **OK** to accept the variable.
8. To disconnect a variable from an object, click the **Disconnect** button. The Variable Navigator remains open, allowing you to make a new variable selection.

Navigator Scope

The following table provides information on the different navigator scopes you can use.

Object	Definition
Flowsheet	Provides a list of all available flowsheets in the simulation, so you can select an object from any flowsheet.
Case	Access general case information (for example, about the Main Solver or Optimizer). You can also use this option when a column is the main flowsheet (for example, if you are in a column template, then there are a bunch of variables that have to be accessed through the Case filter. This is because the column template file doesn't have the upper environment, so those variables become available through the case filter).
Basis	Provides a list of property packages or components being used in the case.
Utilities	Provides a list of all available utilities in the simulation. A special utility Object Filter replaces the default filter.

11.20 Simulation Balance Tool

The Simulation Balance Tool command opens the Simulation Balance Tool view. Refer to [Section 7.27 - Simulation Balance Tool](#) for more information.

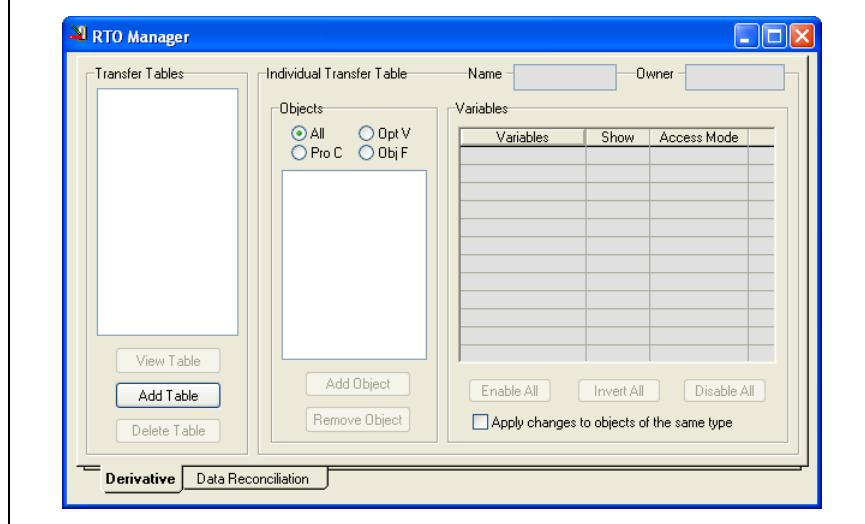
11.21 RTO Manager

The RTO Manager is used for external access to variables of optimization objects for data reconciliation and derivative utilities.

To access the RTO Manager, use the following method:

- Select the RTO Manager command from the Tools menu.

Figure 11.89



All variables used by the RTO Manager are managed through the Transfer Tables. Each transfer table is created for each derivative or data recon utility. Multiple transfer tables can be created for the same utility.

11.21.1 Adding a Transfer Table

Use this procedure to add a transfer table to the RTO Manager.

1. Click the Add Table button to display the Select Utility view.

Figure 11.90

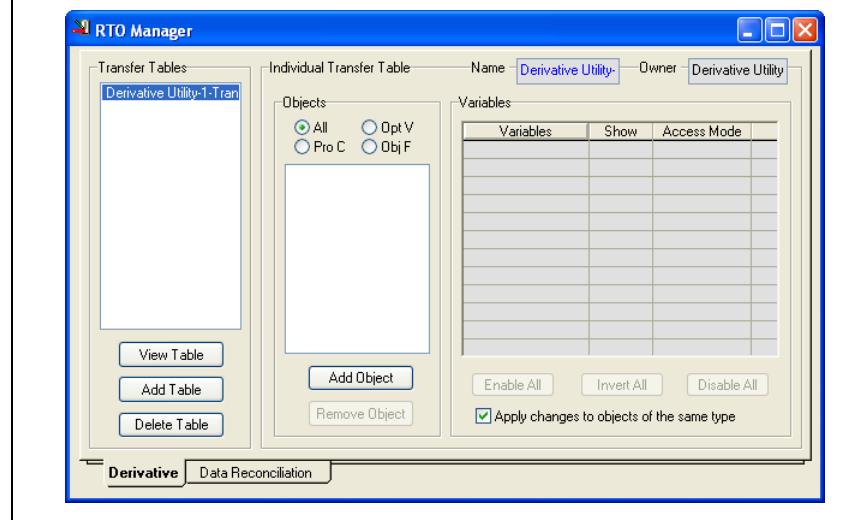


2. From the list of available utilities, select the utility which uses the optimization objects whose variables need external access.

Click the Cancel button at any time to close the Select Utility view without accepting any changes.

3. Click OK. A transfer table which is attached to the selected utility appears in the list of Transfer Tables. If required, type a new name in the Name field.

Figure 11.91



11.21.2 Viewing a Transfer Table

1. From the list of Transfer Tables, click the transfer table you want to view.
2. Click the View Table button. The Transfer Table view appears.
The Transfer Table shows the following for each variable:
 - Object that the variable is attached to.
 - Variable description.
 - Current value.
 - Units of the value.
 - Tag for the variable.

- Access mode of the variable.

Figure 11.92

	Object	Variable	Value	Units	Tag	Access Mode
1	MainStm	stvalue	3402	kg/h	No Tag	Read
2	MainStm	min	2495	kg/h	No Tag	Write
3	KeroProd	stvalue	61.61	m3/h	No Tag	Read
4	KeroProd	max	65.25	m3/h	No Tag	Read/Write
5	MainStm	max	4491	kg/h	No Tag	Write

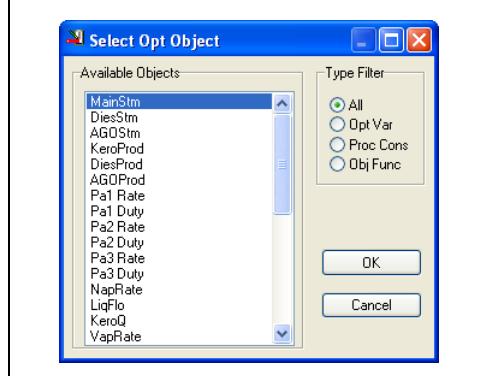
11.21.3 Deleting a Transfer Table

1. From the list of Transfer Tables, click the transfer table you want to delete.
2. Click the Delete button.

11.21.4 Adding Object

1. Click the Add Object button to display Select Opt Object view. Optimization objects which are used by the selected utility will appear in the list of available objects. Type filters can be used to only display objects of a specific type.
1. Click the Add Object button to display Select Opt Object view. Optimization objects which are used by the selected utility will appear in the list of available objects. Type filters can be used to only display objects of a specific type.

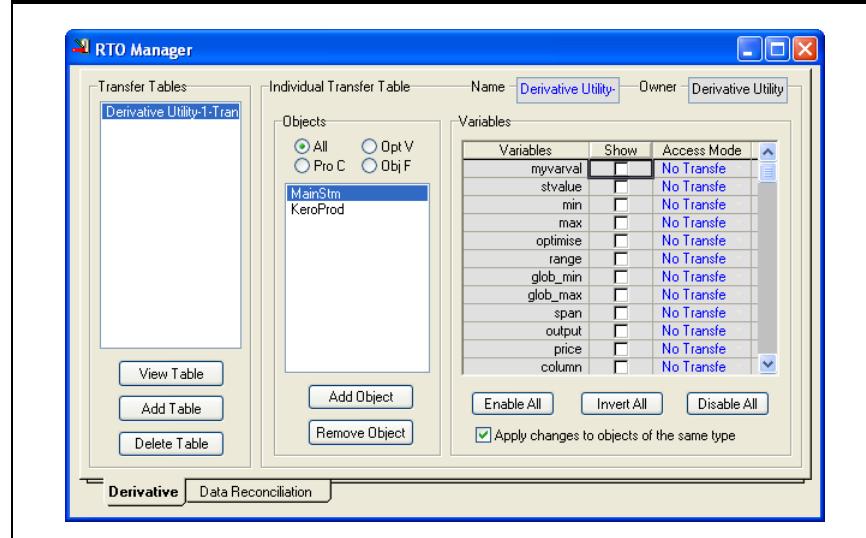
Figure 11.93



2. From the list of available objects, select the object whose variables are needed for external access.

3. Click OK. The selected optimization object will be added to the list of Objects and all variables of the object will also be displayed in the matrix of Variables which includes columns Variables, Show and Access Mode.

Figure 11.94



4. Check the Show checkbox for each variable you want displayed in the transfer table.
5. Click the Enable All button to add all of the variables to the transfer table.
6. Click the Invert All button to check all unchecked checkboxes and uncheck all checked checkboxes.
7. Click the Disable All button to remove all of the variables from the transfer table.
8. If checkbox Apply changes to objects of the same type is checked, any changes in Show and Access Mode will be applied to all other objects of the same type.

11.21.5 Removing Object

1. From the list of Objects, select the object to be removed.
2. Click the Remove Object button. The selected object will be removed from the list of Objects and from the transfer table.

12 Session Preferences

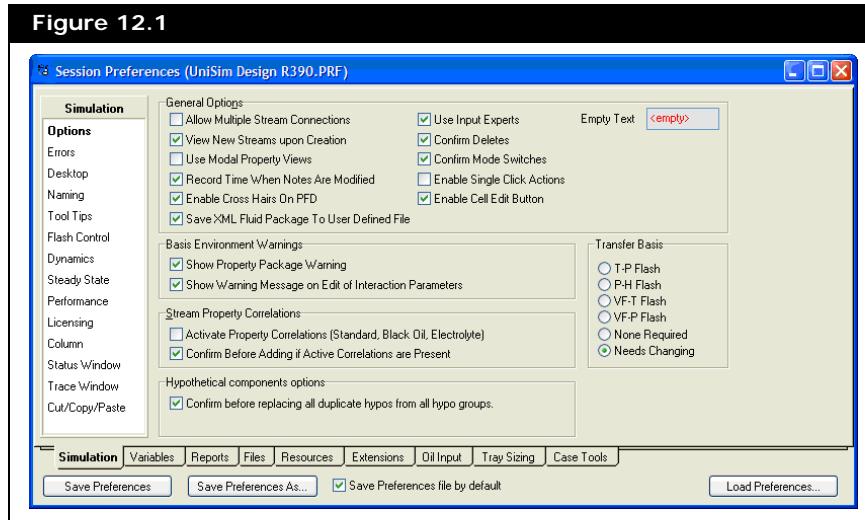
12.1 Introduction	3
12.1.1 Save Preferences	3
12.1.2 Save Preferences As.....	3
12.1.3 Load Preferences.....	4
12.1.4 Save Preferences File by Default.....	4
12.2 Simulation Tab.....	5
12.2.1 Options Page	5
12.2.2 Errors Page	8
12.2.3 Desktop Page	10
12.2.4 Naming Page	11
12.2.5 Tool Tips Page.....	12
12.2.6 Flash Control Page.....	14
12.2.7 Dynamics Page	14
12.2.8 Steady State	16
12.2.9 Performance Page	16
12.2.10 Licensing Page	17
12.2.11 Column Page	18
12.2.12 Status Window Page	18
12.2.13 Trace Window Page	19
12.2.14 Cut/Copy/Paste Page	19
12.3 Variables Tab	20
12.3.1 Units Page	20
12.3.2 Formats Page.....	24
12.4 Reports Tab	26
12.4.1 Format/Layout Page	26
12.4.2 Text Format Page	27
12.4.3 Datasheets Page	28
12.4.4 Company Info Page	29
12.5 Files Tab	30
12.5.1 Options Page	30
12.5.2 Locations Page.....	31
12.5.3 Revision Control.....	32
12.5.4 Databases	34
12.6 Resources Tab	36
12.6.1 Colours Page	36

12.6.2 Fonts Page	37
12.6.3 Icons Page	38
12.6.4 Cursors Page	40
12.6.5 Sounds Page.....	41
12.7 Extensions Tab	42
12.8 Oil Input Tab	43
12.8.1 Assay Definition Page	43
12.8.2 Assay Options Page	44
12.9 Tray Sizing Tab	44
12.9.1 Parameters Page	45
12.9.2 Packed Page	46
12.9.3 Trayed Page.....	46
12.10 Case Tools Tab.....	48
12.10.1 Scenario Manager.....	48
12.10.2 Simulation Balance Tool.....	49

12.1 Introduction

The Session Preferences view is used to specify default information for the simulation case. This information includes Automatic Naming Formats, Units, Colours, Fonts, Icons, etc. Multiple Session Preferences can be saved for use in other simulations.

Access the Session Preferences view by selecting the Preferences command from the Tools menu in any UniSim Design environment.



There are a number of tabs associated with the view. Common to all tabs are the three buttons along the bottom of the view:

- Save Preferences
- Save Preferences As...
- Save Preferences file by default
- Load Preferences...

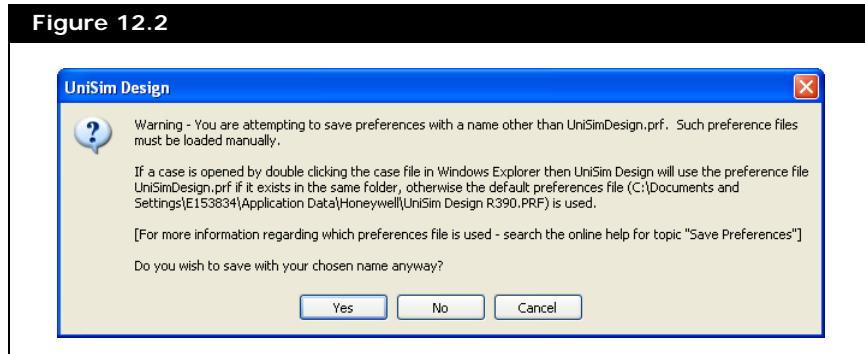
12.1.1 Save Preferences

1. Click the **Save Preferences** button. The current preferences are saved.

12.1.2 Save Preferences As...

1. Click the **Save Preferences As...** button. The Save Preference File view appears.
2. Specify the name and location for your preference file.

3. Click **Save**. You will be prompted with the following message:



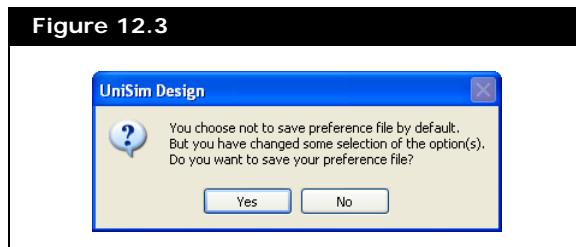
Click **Yes**, **No** or **Cancel** to continue.

12.1.3 Load Preferences...

1. Click the **Load Preferences...** button. The Open Preferences File view appears.
2. Browse to the location of your preference file (*.prf).
3. Select the file you want to load and click **Open**.

12.1.4 Save Preferences File by Default

1. Check on the **Save Preferences file by default** check box. The preference file will be automatically saved onto user's machine when exiting the UniSim Design application.
2. Check off the **Save Preferences file by default** check box. The preference file will not be saved automatically unless user has changed some setting in the preference file. A message box as follow will appear when exiting the UniSim Design Application.



The user can decide if there is a need to save the preference file on to their machine by clicking Yes or No.

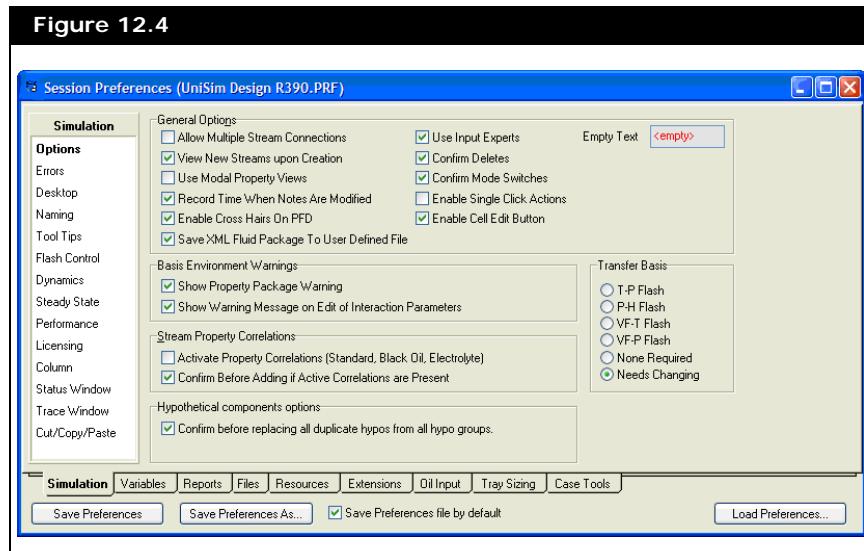
12.2 Simulation Tab

The Simulation tab contains the following pages:

- Options
- Errors
- Desktop
- Naming
- Tool Tips
- Flash Control
- Dynamics
- Steady State
- Performance
- Licensing
- Column
- Status Window
- Trace Window
- Cut/Copy/Paste

12.2.1 Options Page

The Options page contains five groups: General Options, Basis Environment Warnings, Stream Property Correlations, Hypothetical components options and Transfer Basis.



The following table lists and describes the checkboxes in the General

Options group.

Option	Description
Allow Multiple Stream Connections	Controls whether lists of streams are filtered only to those that are not currently connected. If this checkbox is unchecked, when you use the drop-down list of streams to select a feed to an operation, only streams that are not already connected as a feed to an operation appear in the list. If the checkbox is checked, all the streams in the flowsheet appear, including the ones that you cannot connect as feed streams.
View New Streams Upon Creation	If checked, the property view for the stream automatically appears when you add a new stream.
Use Modal Property Views	When checked, all property views appear as modal (with a Pin). When views are modal, you can individually make each property view non-modal by clicking the Pin in the upper corner of the view.
Record Time When Notes are Modified	When checked, all notes are time stamped when they are modified.
Enable Cross Hairs on PFD	When checked, any time the cursor is positioned over the PFD view, a set of vertical and horizontal lines appear and intersect where the cursor point is in the view. When unchecked, the set of lines do not appear.
Save XML Fluid Package to User Defined File	When this checkbox is checked, you can save and export an XML file that contains a fluid package to the user-defined file so that HYPROPIII can manually read the file.
Use Input Experts	Column operations have an optional installation expert built in to assist you in the installation. When this checkbox is checked, the Input Expert guides you through the Column installation.
Confirm Deletes	When checked, you are prompted for confirmation before deleting an object. If the checkbox is unchecked, the object is deleted when the instruction is given. It is recommended that you keep this option checked.
Confirm Mode Switches	When this checkbox is checked, you are prompted for confirmation when changing to or from Dynamics mode.
Enable Single Click Actions	When checked, all objects that require a double-click only require a single-click.
Enable Cell Edit Button	When checked, a cell that is 'editable' has a button appearing on the left side when the cell has focus. Clicking this button, accesses the cell's edit functions (similar to pressing F2).

The Basis Environment Warnings group consists of two checkboxes:

- **Show Property Package Warning** checkbox. When checked the warning message related to the property package will be displayed.
- **Show Warning Message on Edit of Interaction Parameters** checkbox to show the related warning.

The Stream Property Correlations group has two checkboxes:

- **Activate Property Correlations** checkbox

When checked, UniSim Design will activate all available Black Oil, Electrolyte, and Standard Property Correlations. The activation process does not guarantee that each stream in the case will contain all active correlations.

Before the UniSim Design Correlation Manager appends any stream correlations it first checks each streams Fluid Package to confirm that the required information will be available to that correlations calculation. If that stream fluid does not supply the required information the correlation cannot be appended. If the checkbox is not selected the UniSim Design Correlation Manager will not activate any new property correlations and will not remove any correlations previously added. If this checkbox is unselected and then reselected, the Correlation Manager will repeat the process of activating and appending all available Black Oil, Electrolyte, and Standard Property Correlations.

- **Confirm Before Adding if Active Correlations are Present** checkbox

When checked, a warning message will appear when UniSim Design activates a correlation in a case which already contains some active correlations. This warning message asks you if you want to add a full list of correlations in addition to the existing correlations.

The Hypothetical components options group checkbox:

- **Confirm before replacing duplicate hypos from all hypo groups**

Select Confirm before replacing all duplicate hypos from all hypo groups checkbox to have some options on replacing duplicate hypos. If this checkbox is not checked, all duplicate hypos will be replaced without warning by their existing counterparts when the Translocate button is activated in the Hypotheticals tab of the Simulation Basis Manager view.

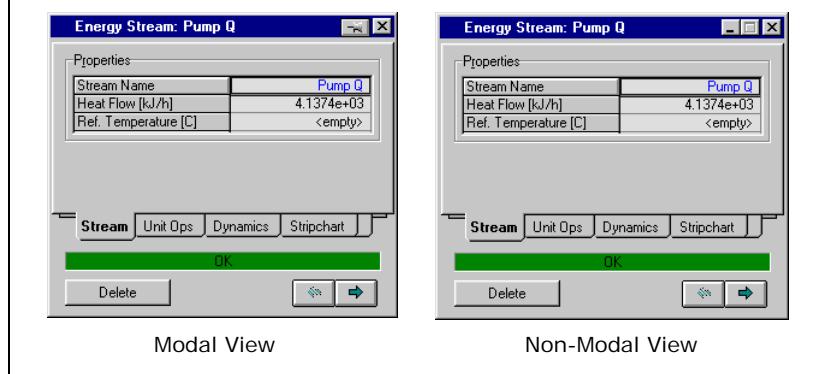
The Transfer Basis group has six checkboxes. Use the checkboxes to select the default transfer basis to be used in stream cutters, subflowsheet templates, column internal/external stream connections, etc.

Modal vs. Non-Modal Property Views

When a view is modal, you cannot access any other element in the simulation (i.e., 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 unnecessary views.

Figure 12.5



A modal view with a Pin can be converted to a non-modal view by clicking the Pin icon.

The modal view is indicated by the substitution of the Minimize/Maximize icons with a Pin icon. Some modal views, such as the Input Composition view, do not have Pins. Click the Pin icon to switch to a non-modal view.

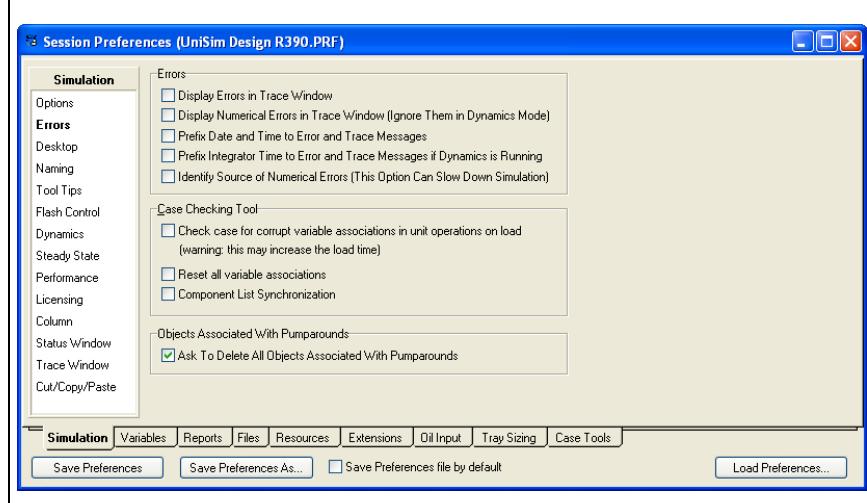
Non-modal views are not restrictive because you can leave a non-modal view open and interact with any other view or menu item. The non-modal view has the Minimize/Maximize icons in the upper right corner of the view.

12.2.2 Errors Page

Refer to [Section 1.3 - Object Status Window/Trace Window](#) for more information on the Trace Window.

The Errors page contains three groups: Errors, Case Checking Tool and Objects Associated with Pumparounds.

Figure 12.6



The **Errors** group has five checkboxes which, when checked, send the specified errors to the Trace Window. When these checkboxes are unchecked, you are not prompted to acknowledge errors.

The following is a brief description of each error type.

Error Type	Description
Display Errors in Trace Window	The basic error message is displayed within the trace window rather than via a pop-up window.
Display Numerical Errors in Trace Window	This option sends numerical errors to the trace window rather than via a pop-up error message window. In dynamics mode, checking this option ignores numerical errors so the integrator will continue to run even if the numerical errors occur.
Prefix Date and Time to Error	The date and time (according to the clock and calendar of your CPU) is placed before the error message.
Prefix Integrator Time to Error	The integrator time is placed before the error message (this only applies when the integrator is running in a dynamics case).
Identify Source of Numerical Errors	This option will try to identify the source of a numerical error and display it within the trace window. This can slow UniSim Design down so it should only be used when you are trying to track down errors to clean them up. Once you have finished tracking errors, turn this option off. This option may not be able to identify the source if it does not occur within a unit operation (i.e., If the error was caused by a strip chart).

The **Case Checking Tool** group has three checkboxes.

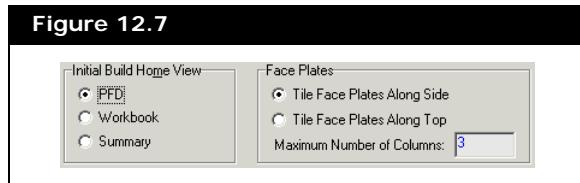
- **Check case for corrupt variable associations in unit operations on load (warning: this may increase the load time).** When this box is checked USD will do a search for all the variables associated with the unit ops during case loading. If a variable is found to be corrupted, USD will do a correction automatically. This may prevent some cases from crashing in loading.
- **Reset all variables associations.** When this is checked, USD will reset all variables association with unit op unconditionally during case loading. The benefit is that it may resolve some mysterious crash of an old case. The penalty is that all the variables "Calculated By" information will be lost. It will be reestablished in the next Flowsheet solution.
- **Component List Synchronization.** When this box is checked on, USD will do a check and synchronize all the Fluid Packages associated Component Lists with the Master Component List during case loading. Some old cases may have this issue.

The **Objects Associated With Pumparounds** group has one checkbox **Ask to Delete All Objects Associated With Pumparounds**. When user deletes/removes a column Pumparound object, a message will pop up to confirm with user if he/she wants to keep or delete all the objects associated with the Pumparound object if this check box is checked on. Otherwise, the objects associated with the Pumparound object will be

kept in the flowsheet.

12.2.3 Desktop Page

The Desktop page contains two groups: Initial Build Home View and Face Plates.

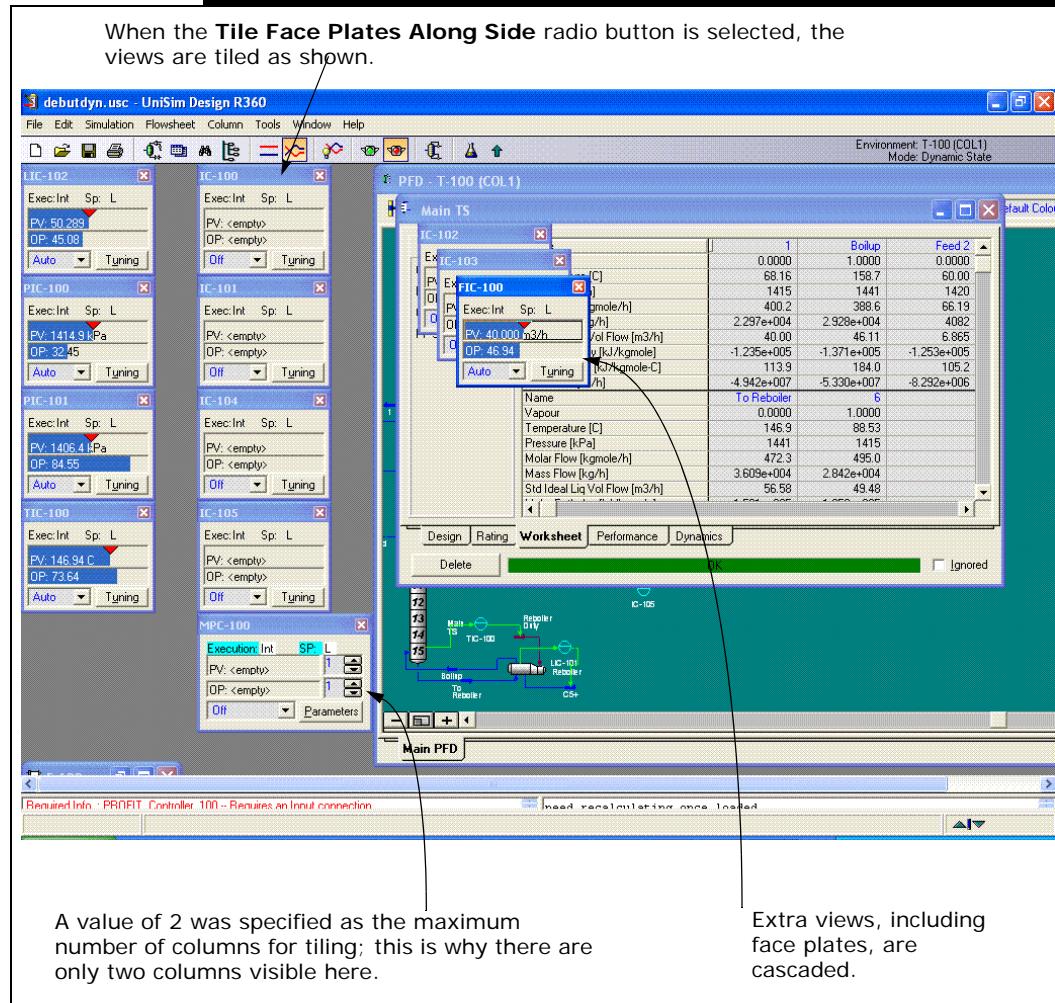


The Initial Build Home View group contains radio buttons that let you specify which of the three main views (Workbook, PFD, or Summary) automatically appears when you first enter the Simulation environment. This does not restrict your access within that environment, as the Workbook, PFD, and Summary views can all be open at the same time. This setting only establishes the view that appears 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 click the Arrange Desktop command in the Window menu, the

face plates are organized according to your specifications in this group.

Figure 12.8



The face plates are either 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 selected radio button) in the Maximum Number of Columns/Rows field. Any excess face plates that cannot be placed on the Desktop due to the columns/rows limit are cascaded with other open views.

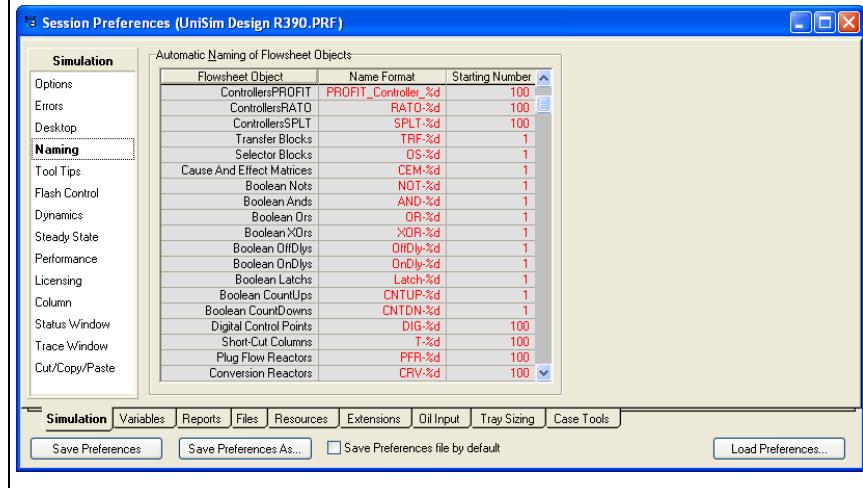
12.2.4 Naming Page

There are no restrictions in naming streams and operations. You can use more than one word, and spaces are allowed.

The Naming page dictates how streams and operations are named when they are installed. You can specify the naming convention for each type of operation. For each flowsheet object, specify a naming convention and a starting number.

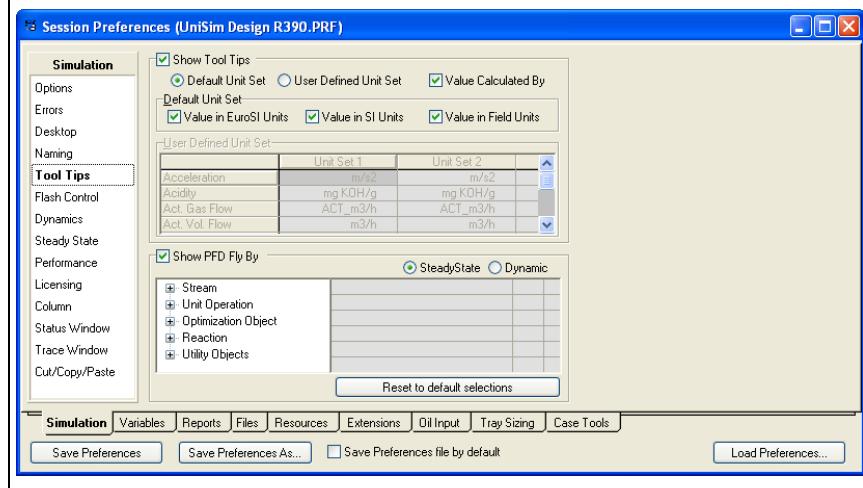
For example, in the figure below, Energy Streams are indicated as Q-%d, with a starting number of 100. The first energy stream installed in the simulation is named Q-100, the second Q-101, and so on. The automatic naming function is provided for convenience. You can change any default name at any time within the flowsheet.

Figure 12.9



12.2.5 Tool Tips Page

Figure 12.10



The following table lists and describes the objects in the **Show Tool**

Tips group.

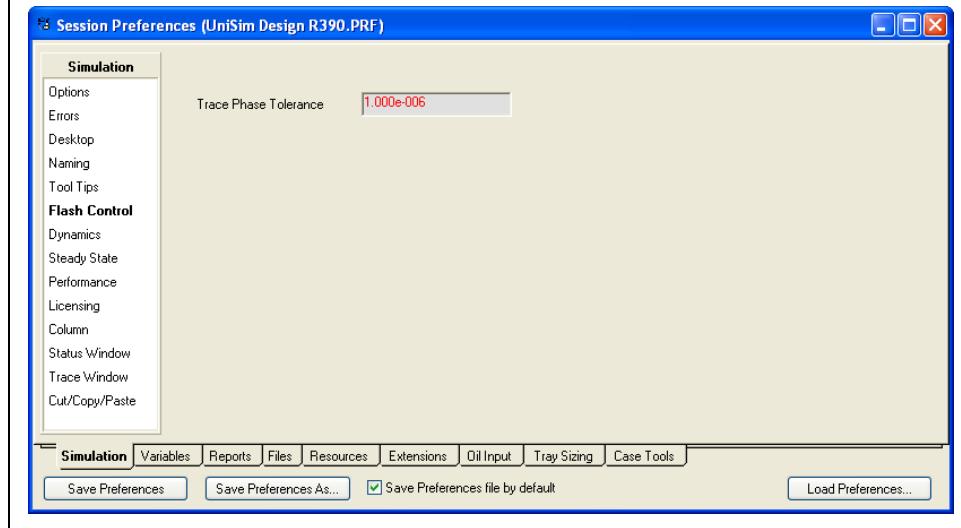
Object	Description
Show ToolTips checkbox	Activates the available tool tips. When this checkbox is checked you can use the checkboxes in the ToolTips group.
Default Unit Set radio button	Select this radio button to use the default unit set.
User Defined Unit Set radio button	Select this radio button to define a custom unit set. The User Defined Unit Set table becomes active, and you can define the unit set.
Value Calculated By checkbox	Displays what operation calculated the value in the tool tip. Uncheck the checkbox if you do not want to see this value.
Value in EuroSI Units checkbox	Displays the value in European SI units in the tool tip. Uncheck the checkbox if you do not want to see this value.
Value in Field Units checkbox	Displays the value in Field units in the tool tip. Uncheck the checkbox if you do not want to see this value.
Value in SI Units checkbox	Displays the value in SI units in the tool tip. Uncheck the checkbox if you do not want to see this value.
User Define User Set table	Available only when the User Defined Unit Set radio button is selected. Allows you to define the unit set.

The following table lists and describes the objects in the **Show PFD Fly By** group.

Object	Description
Show PFD Fly By checkbox	Click this checkbox to see the Fly by text in the PFD. The Fly by displays information about an object when you move the cursor over it in the PFD.
Steady State radio button	Select this radio button to view the Fly by text in Steady State mode.
Dynamic State radio button	Select this radio button to view the Fly by text in Dynamic Mode.
PFD object tree browser	Select a PFD object type from the tree browser. A list of Fly by options appears in the table to the right of the tree browser.
Show PFD Fly By Table	Select the checkboxes in the table to select the parameters you want to view in the Fly by text. By default, Temperature, Pressure, and Molar Flow are already selected. Some options are available only in Dynamic mode.
Reset to default selections button	Click this button to reset the Show PFD Fly By options back to the default setting.

12.2.6 Flash Control Page

Figure 12.11



Trace Phase Tolerance is the threshold value for phase fraction; it is used in the flash calculations. If phase fraction value of a phase drops below the Trace Phase Tolerance value during flash calculation, then that phase is deleted.

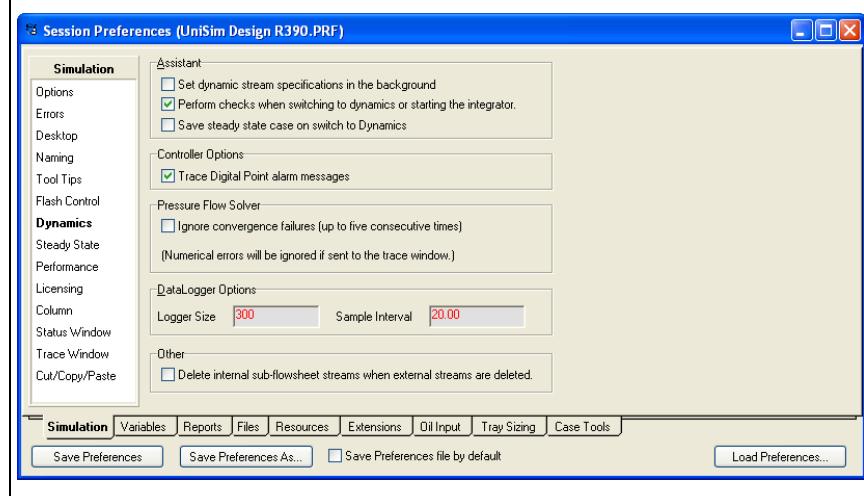
The Default Trace Phase Tolerance value is 1E-6. This value can be changed by the user. Any user entered Trace Phase Tolerance value should be between 1E-6 and 1E-12.

12.2.7 Dynamics Page

There are five groups in the Dynamics page: Assistant, Controller

Options, Pressure Flow Solver, DataLogger Options and Other.

Figure 12.12

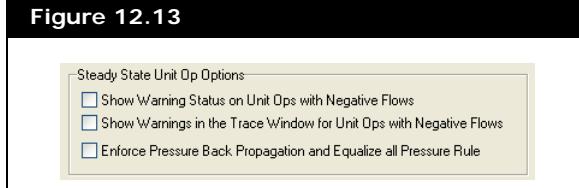


The following table lists and describes the objects on the Dynamics page.

Object	Description
Set dynamic stream specifications in the background checkbox	Check the checkbox to automatically set all of the dynamic stream specifications.
Perform checks when switching to dynamics or starting the integrator checkbox	Check the checkbox if you want to use the Dynamics Assistant every time you switch from steady state to dynamics, or when starting the Integrator.
Save steady state case on switch to Dynamics	Check the checkbox if you want to save the steady state case before changing to dynamic state.
Trace controller alarm messages checkbox	Check the checkbox if you want to see controller alarm messages in the Trace Window.
Ignore convergence failures checkbox	When checked, convergence failures for up to five pressure flow steps are ignored. Checking this option is not recommended, but it can be useful when you know a converging case converges while having problems during one of the calculation steps.
Logger Size field	Specify the amount of data (steps) you want stripcharts to keep.
Sample Interval field	Specify how frequent UniSim Design records the data for the stripcharts. The shorter the amount of time, the more frequent UniSim Design records the data.
Delete Internal sub-flowsheet streams when external streams are deleted checkbox	When checked, all internal sub-flowsheet streams are deleted when the corresponding external stream is deleted.

12.2.8 Steady State

Figure 12.13



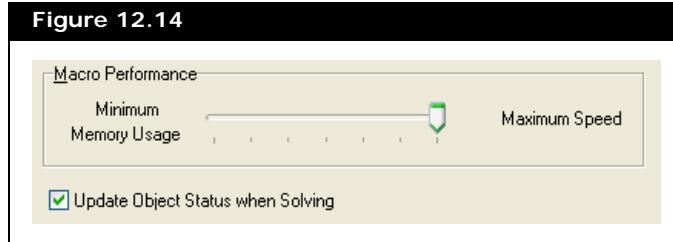
There are three checkboxes in the **Steady State Unit Op Options** group:

- **Show Warning Status on Unit Ops with Negative Flows.**
- **Show Warnings in the Trace Window for Unit Ops with Negative Flows.**
- **Enforce Pressure Back Propagation and Equalize all Pressure Rule.** When this is checked, unit op such as 3 Phase Separator, Separator, Tank, Single Outlet Vessel, CSTR, Gibbs Reactor, Equilibrium Reactor and Conversion Reactor will conduct back pressure calculation and equalize all pressure for multiple feed streams when pressure value is not provided by user.

12.2.9 Performance Page

The Performance page allows you to manipulate the UniSim Design computation time and calculation process.

Figure 12.14



Macro Performance Group

The Macro Performance group contains the Macro Performance slider. Move the slider to your required setting to balance the Macro speed with your memory requirements. The faster the setting, the more memory your computer requires to run the macros.

The default setting for the **Update Object Status when Solving** checkbox is selected.

Update Object Status when Solving Checkbox

When the Update Object Status when Solving checkbox is selected, the object status services updates each time an object is solved. For

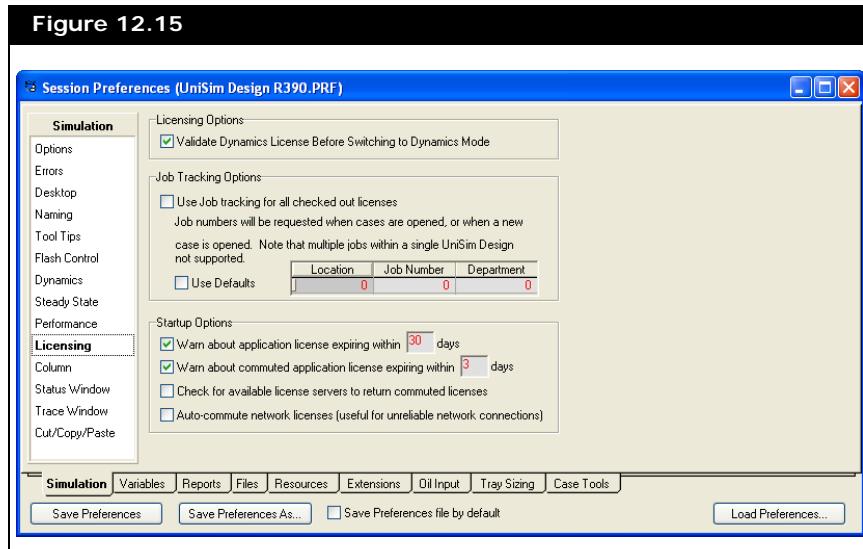
example, as UniSim Design solves the entire flowsheet, the object status goes between “not solved” and “solved” as UniSim Design encounters not solved objects and solved objects in the flowsheet.

When the checkbox is clear, UniSim Design only updates the object status of all the objects in the flowsheet at the end of the solve pass, in other words, when UniSim Design has finished solving the entire flowsheet. The calculation time decreases and performance improves when you clear the checkbox.

12.2.10 Licensing Page

To purchase additional licenses or products, contact Honeywell, or e-mail us at
unisim.support@honeywell.com.

The Licensing page contains the following groups: Licensing Options, Job Tracking Options and Startup Options.



The default licensing behaviour of the UniSim Design Dynamics license can be changed in the Licensing Options group.

The options in the Job Tracking Options group are Use Job tracking for all checked out licenses and Use Defaults.

- If you check the **Use Job tracking for all checked out licenses** checkbox, job numbers will be requested when either a new or old case is opened. This option does not support multiple jobs within a single UniSim Design session.
- If you check the **Use Defaults** checkbox, you can enter defaults values in the adjacent table. If these defaults are set, UniSim Design will not ask for any job information.

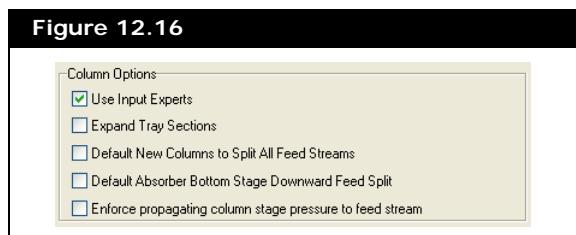
The options in the **Startup Options** group.

- You have the option to change the warning about when your license will expire. The default is 30 days.

- You have the option to change the warning for the number of days that a commuted license will expire.
- You have the option to check for available license servers to return commuted licenses
- You will have the ability to choose to auto-commute a network license.

12.2.11 Column Page

The Column page consists of one group: Column Options.



The Column Options group contains the following checkboxes:

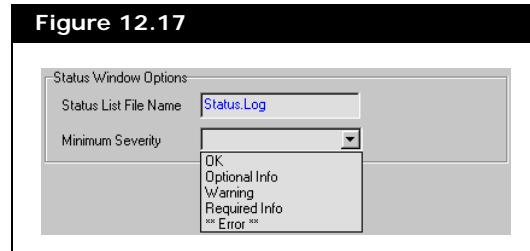
- **Use Input Experts.** Column operations have an optional input expert built in to assist you in the installation. When this checkbox is checked, you are guided through the Column installation.
- **Expand Tray Sections.** When checked, it shows a fully expanded column in the Column environment PFD. When the checkbox is unchecked, the column appears with the minimum required number of trays (those trays which have streams [inlet or outlet] attached to them).
- **Default New Columns to Split All Feed Streams.** When checked, all feed streams in any newly created column are automatically split. (The option does not affect existing columns and any feed splitting options already selected.)
- **Default Absorber Bottom Stage Downward Feed Split.** When checked, the liquid phase of any feed stream with "Split" checked to the bottom stage of any newly created absorber will flow to the sump separator below the bottom tray of the absorber, while the split vapour phase in the stream will flow to the specified tray, bottom tray. (The option does not affect existing columns and any feed splitting options already selected.)
- **Enforce propagating column stage pressure to feed stream.** When checked, column stage pressure will be propagated to the feed stream which is connected to the stage.

The options in the Status Window page are exactly the same as the options offered in the Status List Properties view. See the **Object Status Window** sub-section from **Section 1.3.3 - Object Inspect Menu** for more information.

12.2.12 Status Window Page

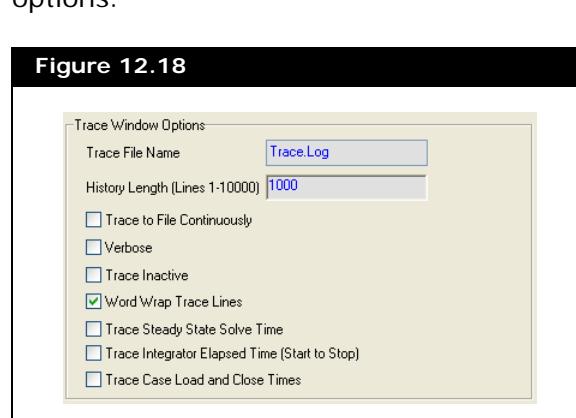
The Status Window page allows you to manipulate the file that saves

the status message that appears in the Status Window.



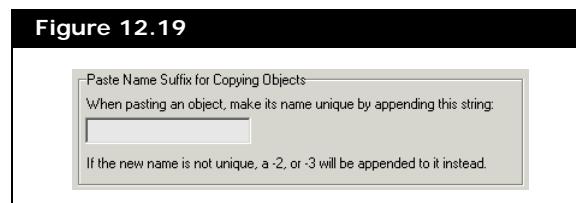
12.2.13 Trace Window Page

The options in the Trace Window page is exactly the same as the options offered in the Trace Properties view. See **Trace Window** subsection from the **Section 1.3.3 - Object Inspect Menu** for more information.



12.2.14 Cut/Copy/Paste Page

The Cut/Copy/Paste page allows you to add specific extension name to a copied unit operation after you have paste the operation.



For example, if you specified **-optim** as the extension to the copied

name, as shown in the figure below:

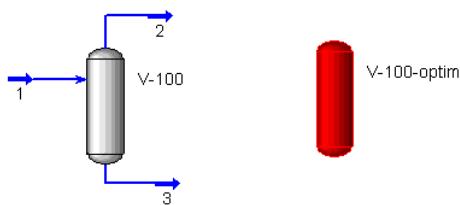
Figure 12.20

Paste Name Suffix for Copying Objects
When pasting an object, make its name unique by appending this string:

If the new name is not unique, a -2, or -3 will be appended to it instead.

The copied and paste object will have **-optim** added to the back of its name, as shown in the figure below of a copied separator:

Figure 12.21



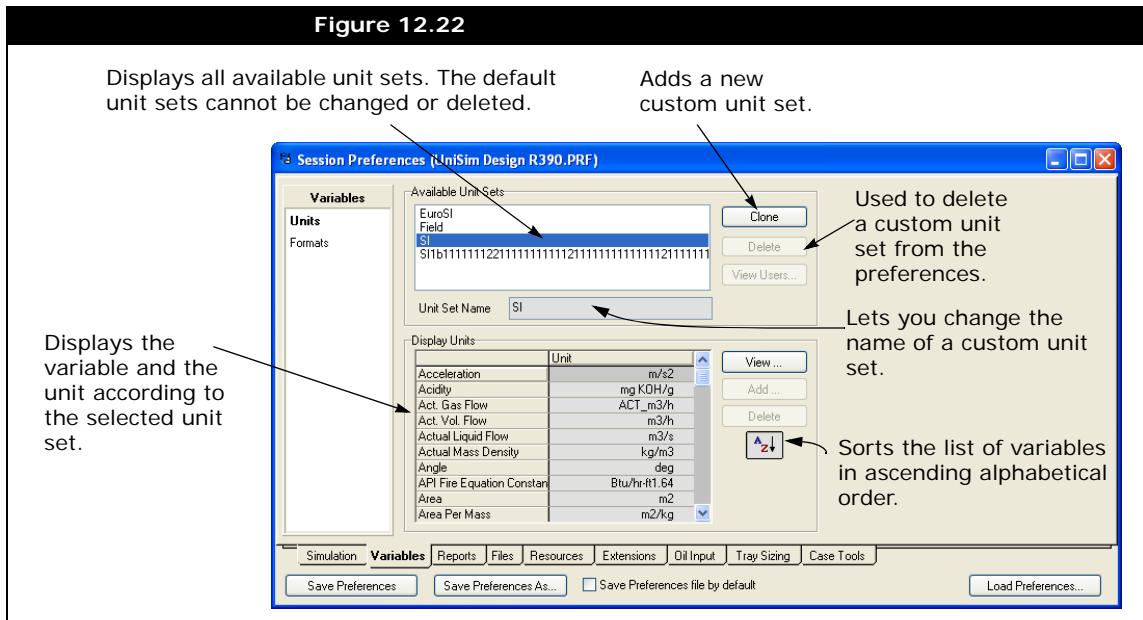
12.3 Variables Tab

The Variables tab has two pages: Units and Formats.

12.3.1 Units Page

The Units page allows you to select and manipulate the unit set used in

the simulation case.



UniSim Design contains the following default unit sets:

- Field
- SI
- EuroSI

These three sets cannot be modified in any way or deleted. If you want UniSim Design to display information in units other than the default, you can create your own custom unit sets.

Adding a Unit Set

A custom unit set lets you mix SI and Field units within the same unit set.

1. Select the unit set you want to clone from the list of available unit sets.
2. Click the **Clone** button. A new unit set appears with the name NewUser.
3. Change the default name in the Unit Set Name field, if required.

The units used in the new unit set are the same as the unit set you cloned.

Deleting a Unit Set

You can not delete the three default unit sets: SI, Field, and Euro SI.

1. Select the unit set you want to delete from the list of available unit sets.

sets.

2. Click the **Delete** button.

You are not prompted to confirm the deletion of a unit set, so ensure that you have selected the correct unit set to delete.

Changing the Units in a Unit Set

You can not modify the units in any of the three default unit sets: SI, Field, and Euro SI.

This selection does not change the UniSim Design internal unit set.

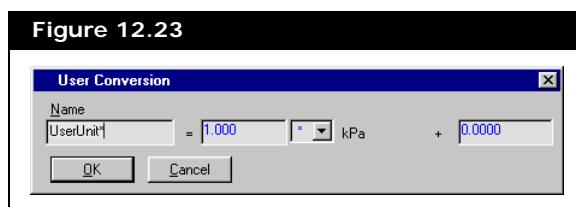
The UniSim Design internal unit is always displayed on the User Conversion view.

1. From the list of available unit sets, select the user-defined unit set you want to use for your simulation.
2. In the Displayed Units group, click the unit you want to change (e.g., Temperature).
3. In the selected unit cell, press the space bar to open a drop-down list. The drop-down list shows all available convertible units for that unit type. For Temperature, you see C, K, F, and R and any user conversions that were created.
4. From the list, select the unit you want use.

Adding a Unit Conversion

If you require a unit that is not available in the UniSim Design database, you can create your own unit and supply a conversion factor. You can only add a unit to a user defined unit set.

1. From the list of available unit sets, select the customized unit set you want to use for your simulation.
2. In the Displayed Units group, select the unit for which you want to add a conversion.
3. Click the **Add** button to display the User Conversion view.



4. In the **Name** field, type the name of your new unit.
5. In the second field (the multiply/divide field), type the conversion factor between your unit and the UniSim Design internal unit.
6. From the drop-down list, specify whether you want to multiply or divide by the conversion factor.
7. In the final field (the add/subtract field), type the conversion factor between your unit and the UniSim Design internal unit.

To add a factor, specify a value in this field. To subtract, place a

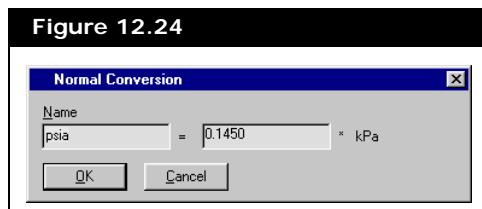
negative sign in front of the number (for example, enter -2.0).

8. Click **OK**. The User Conversion view closes with this unit as the active unit for that unit type.

Viewing a Unit Conversion

The View button lets you view the conversion factor used to convert a unit from its internal unit (SI) to the selected user defined unit in the unit set.

1. From the list of available unit sets, select the unit set you want to use for your simulation.
2. In the Displayed Units group, select the unit for which you want to view the conversion.
3. Click the **View** button. The Conversion view appears.



4. Do one of the following:
 - Click the **OK** button to accept any changes made to a user unit conversion and close the view.
 - Click the **Cancel** button to close the view without accepting any changes.

Deleting a Unit Conversion

You can only delete user defined unit conversions.

1. From the list of available unit sets, select the required unit set.
2. In the Displayed Units group, select the user-created unit you want to delete.
3. Click the **Delete** button. You are not prompted to confirm the deletion of the unit. The unit returns to the UniSim Design default unit.

Viewing Users of a Unit Set

Units sets can be attached to spreadsheets and reports, so spreadsheets can have different unit sets than your simulation.

1. From the list of available unit sets, select the unit set you want to use for your simulation.

Clicking the **OK** button closes the view and returns you to the Session Preferences view.

- Click the **View Users** button. The Unit Set view appears. This button is available only if the unit set is attached to an object.

Figure 12.25

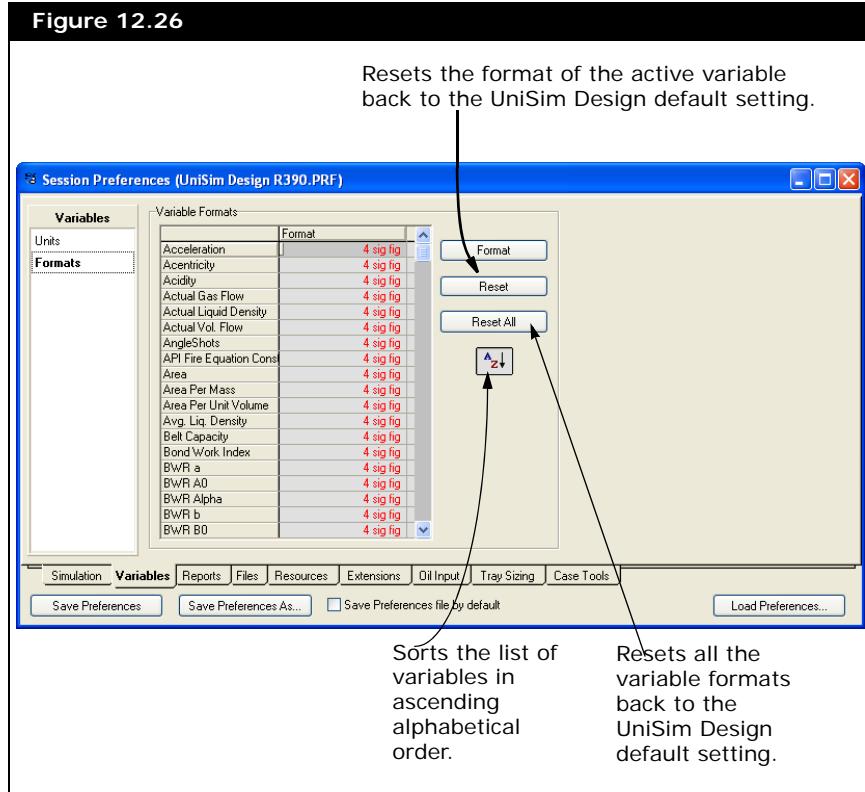


- From the list of available users, select the user you want to view.
- Click the **View selected object** button. The property view of the selected user appears and the Unit Set view closes.

12.3.2 Formats Page

This page lets you specify how variables appear.

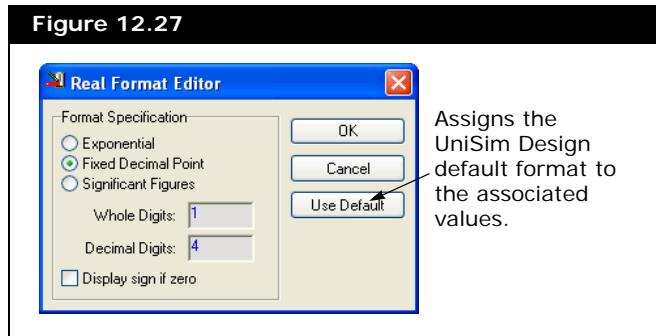
Figure 12.26



Modifying Variable Formats

To select more than one format at a time, hold down the **CTRL** key, and click each variable you want to select.

1. From the list of variable formats, click the format cell of the variable you want to modify.
2. Click the **Format** button. The Real Format Editor view appears.



3. Select the format you want to use for your variable.
4. Click **OK** to accept the changes and close the view.

Real Format Editor

The Real Format Editor lets you set the format for displaying values in UniSim Design.

The following table lists and describes the available formats:

Radio Button	Description
Exponential	Displays the values in scientific notation. The number of significant digits after the decimal point is set in the Significant Figure field. For example: <ul style="list-style-type: none">• Entered or calculated value: 10000.5• Significant Figure: 5 (includes the first whole digit)• Final display: 1.0001e+04

Radio Button	Description
Fixed Decimal Point	<p>Displays the values in decimal notation. The number of whole digits and significant digits after the decimal point are set in the Whole Digits and Decimal Digits fields. For example:</p> <ul style="list-style-type: none"> • Entered value: 100.5 • Whole digits: 3 • Decimal digits: 2 • Final display: 100.50 <p>If the entered or calculated value exceed the specified whole digits, UniSim Design displays the value as Exponential, with the sum of the specified whole and decimal digits being the number of significant figures.</p> <p>If the Display sign if zero checkbox is checked, UniSim Design displays the sign of the number entered or calculated that is rounded to zero.</p>
Significant Figure	<p>The number of significant digits after the decimal point is set in the Significant Figure field. Displays the values in either decimal notation or scientific notation.</p> <p>Example one:</p> <ul style="list-style-type: none"> • Entered or calculated value: 100.5 • Significant Figure: 5 • Final display: 100.50 <p>Example two:</p> <ul style="list-style-type: none"> • Entered or calculated value: 10000.5 • Significant Figure: 5 • Final display: 1.0001e+04

12.4 Reports Tab

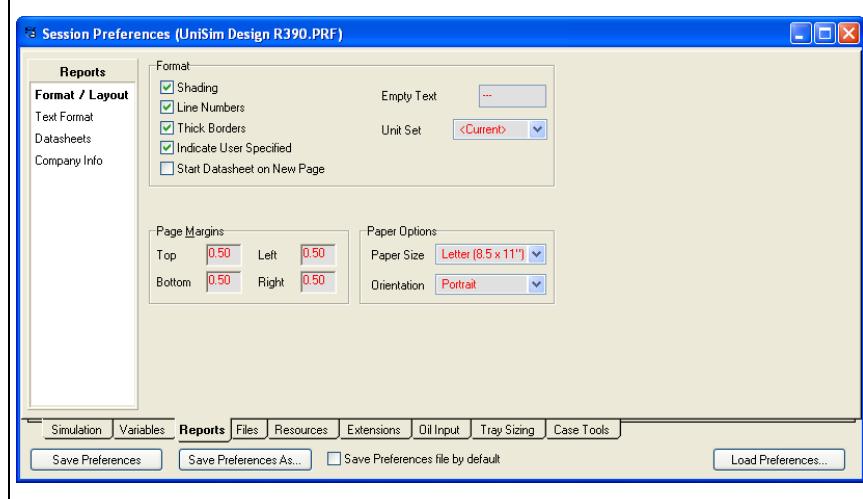
The Reports tab is divided into four pages: Format/Layout, Text Format, Datasheets, and Company Info.

12.4.1 Format/Layout Page

The Format/Layout page provides options for formatting and specifying

the appearance of your printed reports.

Figure 12.28



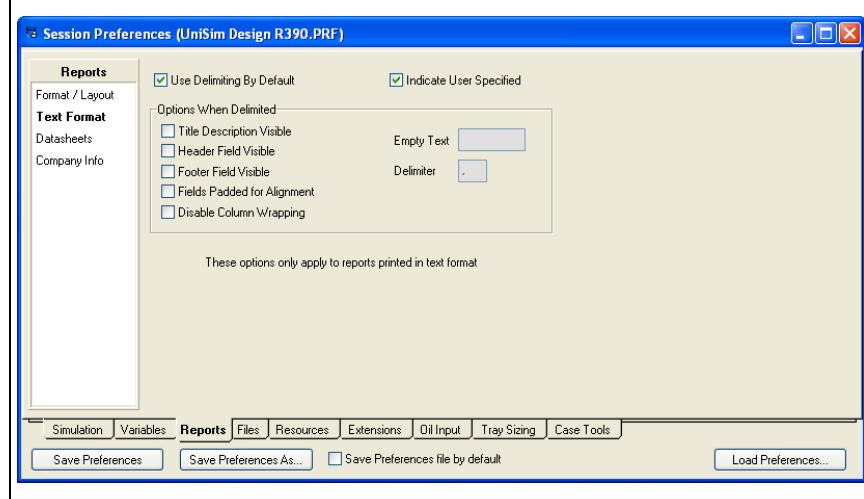
Object	Description
Shading	When checked, headers, footers, and titles are shaded.
Line Number	When checked, line numbers appear on the left side of the report.
Thick Borders	When checked, report border lines are thicker than the other lines in the report.
Indicate User Specified	When checked, any user specified values in the Datasheet are indicated with an asterisk (*).
Start Datasheet on New Page	When checked, each Datasheet starts on a new page.
Empty Text	Specify what you want to display in the Datasheet when there is no value available. The UniSim Design default is "---".
Unit Set	Select the unit set you want your Datasheet to use. This gives you the option of printing Datasheets with different unit sets than your case. For example, your case may be in SI, but you can set the report to appear in Field units.
Page Margins	Set the margins of your page. The values are the distance in inches from the edge of the page.
Paper Options	Select the size of paper on which you want to print. The list contains all the Microsoft defaults.
Orientation	Select the orientation of the data on the paper. You have two options: Portrait or Landscape.

12.4.2 Text Format Page

Use the Text Format page to define the appearance of text in the

report.

Figure 12.29



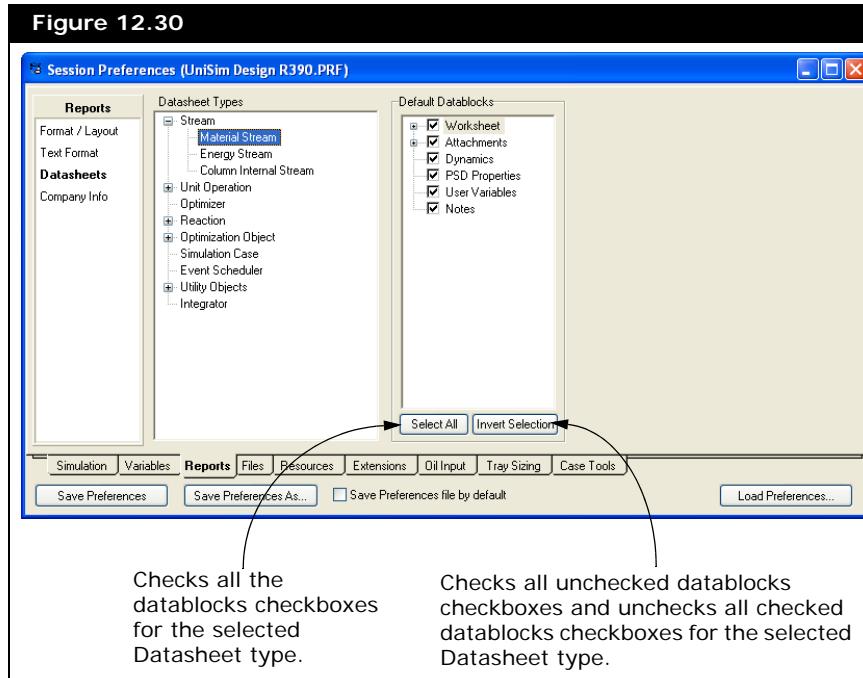
Object	Description
Use Delimiting By Default	Check this checkbox if you want the text file to always be delimited.
Title Description Visible	When checked, a title is added to the text file. The title includes the name of the object and the tabs that are included in the report.
Header Field Visible	When checked, a header is added to the text file. The header includes the company information and the date the report was created.
Footer Field Visible	When checked, a footer is added to the text file. The footer includes the UniSim Design version and build number.
Fields Padded for Alignment	When checked, spaces are added between each field to align the fields.
Disable Column Wrapping	When checked, column wrapping is disabled. This means that text that goes past the edge of the page does not wrap onto the next line.
Empty Text	Specify what you want to display in the Datasheet when there is no value available. The UniSim Design default is "---".
Delimiter	Specify what you want to use as the delimiter in your text file. The UniSim Design default is comma delimited (,).

12.4.3 Datasheets Page

The Datasheets page lets you control which information will appear in

the report for selected streams and operations.

Figure 12.30



12.4.4 Company Info Page

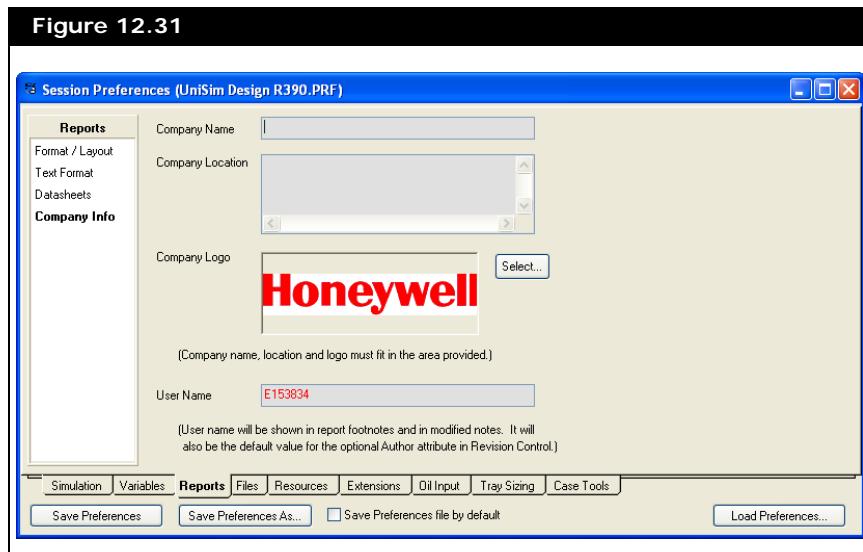
UniSim Design does not automatically resize a bitmap file to fit the logo box on this page.

The size of the sample logo is 4.42 cm wide by 1.88 cm high.

The maximum logo size that can be accommodated by the logo box is 6.55 cm wide by 2.38 cm high.

The Company Info page lets you customize the company information that appears on the report.

Figure 12.31



To modify the company information, do the following:

1. In the Company Name field, type the company name that will appear in the report header.
2. In the Company Location field, type the company location that will appear in the report header.
3. To add a company logo, click the **Select** button. The Open File view appears. Browse to the location of your bitmap file (*.bmp).
4. Select the file you want to import and click the **Open** button.

The logo picture must be in bitmap format.

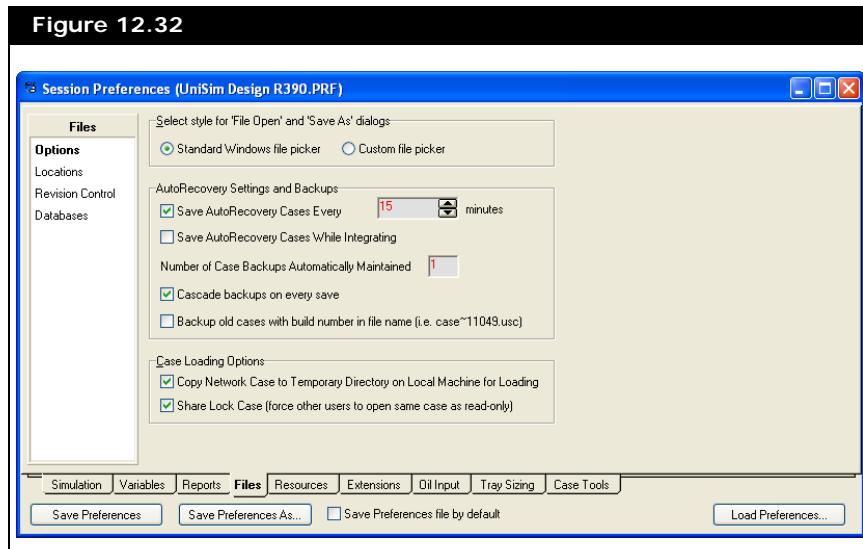
12.5 Files Tab

The Files tab has the following pages:

- Options
- Locations
- Revision Control
- Databases

12.5.1 Options Page

The Options page lets you modify the preferences used when saving a case.



From the Select style for 'File Open' and 'Save As' group, select either the Standard Windows file picker or the custom file picker radio button. The custom file picker displays custom UniSim Design open and save views that show the build in which the case was saved, and the case description (if one was added).

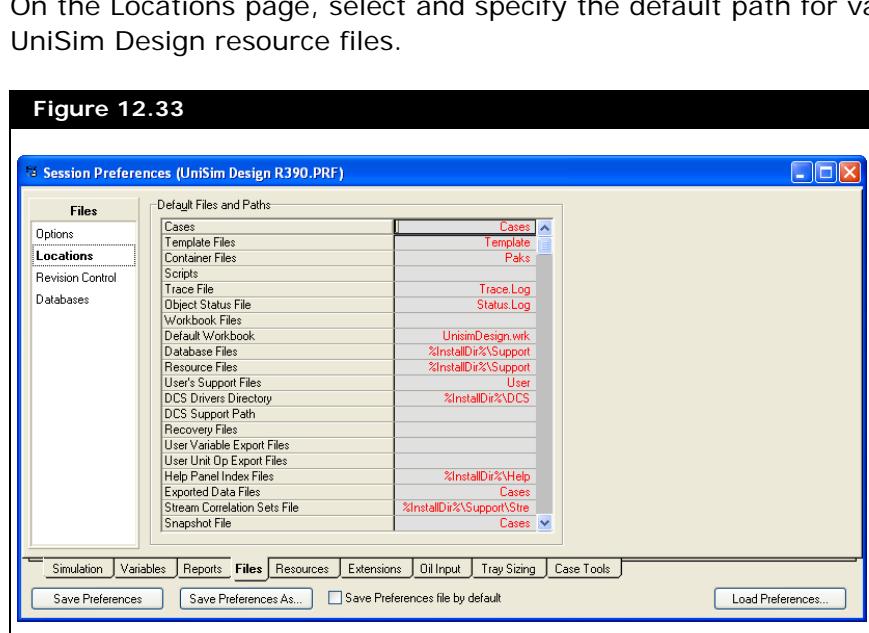
In the AutoRecovery Settings and Backups group, use the checkboxes

to select the options you want to use. The following table lists and describes the checkboxes.

Object	Description
Save AutoRecovery Cases Every	When checked, UniSim Design saves an auto recovery case in the time interval specified in the minutes field. You can use the up and down arrows to increase and decrease the value of the field by one with each click, or you can enter a value directly in the field.
Save AutoRecovery Case While Integrating	When checked, UniSim Design saves an auto recovery case every time the integrator is run. This checkbox is only available when the Save AutoRecovery Cases Every checkbox is checked.
Cascade Backups on Every Save	This checkbox is used in conjunction with the Number of Case Backups Automatically Maintained. When checked, UniSim Design maintains the specified number of backups of each simulation, using the extension bk*. The newest backup is bk0, the next newest bk1, etc.
Backup old cases with build number in file name	Check this box if you want the Backup cases to include the build number.

12.5.2 Locations Page

UniSim Design is set to look in the Cases folder for any case files because the Cases field contains the path "Cases". Leave the field blank if you do not want to use a default location for that option.



In the Default Files and Paths table, specify the file or path in which you want the selected option to reside.

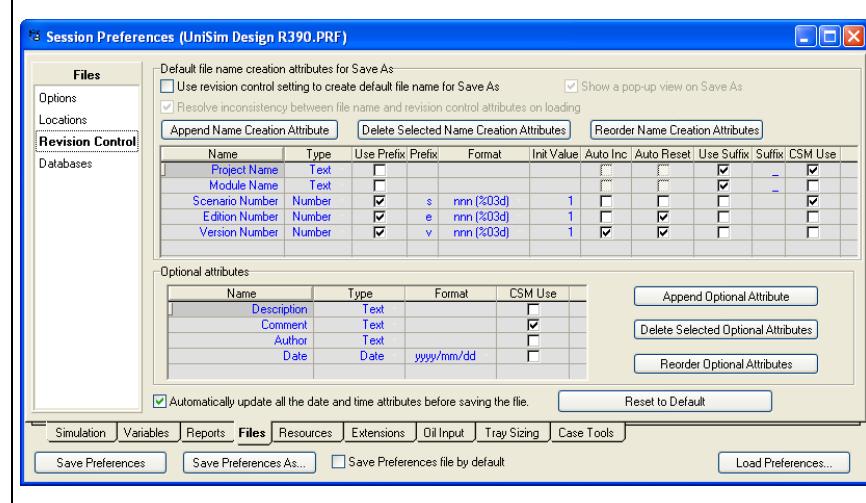
You can scroll down to the Stream Correlation Sets File field and change the default name and path. UniSim Design has access to any custom

correlation set files (xml) which you also have access too.

12.5.3 Revision Control

The Revision Control page lets you set up the preferences for revision control for simulation cases or other types of files. The revision control information can be used to generate appropriate file names and populate optional information related to the file.

Figure 12.34



The name creation attributes are used to automatically create file names for a simulation case or other file type.

The following table describes the items in **Default file name creation attributes for Save As** groupbox.

Object	Description
Show a pop-up view on Save As checkbox	Decide if a pop-up window is to be shown when Save As is invoked. This window allows more options for Save As .
Resolve inconsistency between file name and revision control attributes on loading checkbox	Decides if the inconsistency between the file name and the revision control attributes will be addressed or not after a file is loaded.
Append Name Creation Attribute button	Append a name creation attribute to the attribute list displayed in the matrix below.
Delete Selected Name Creation Attributes button	Delete the selected name creation attributes in the matrix below.
Reorder Name Creation Attributes button	Sort the name creation attributes manually.
Name field	The name of the attribute.

Object	Description
Type Field	The data type of the attribute. The choices are: Text, Number, Date, Time and Yes/No.
Use Prefix Field	Decide if a prefix character is to be used before this field.
Prefix Field	The character used immediately before the content of the attribute.
Format Field	The format for number and date types. The number format is used for generating file name only, but the number itself should not exceed the specified number of digits. In generating the file name, smaller numbers will use leading zeros to make up the number of digits specified.
Init Value Field	The initial or reset value for a number attribute.
Auto Inc Field	Decide if a number is automatically increased by 1 before saving the file. However, if the next default file name will conflict with an existing file in the same directory, the number will keep increasing until a non-conflicting value is found.
Auto Reset Field	Automatically reset to initial value when any attribute above this one changes.
Use Suffix Field	Decide if suffix should be used with the content of this attribute when creating the default file name.
Suffix Field	The character used immediately after the contents of the attribute.
CSM Use Field	Decide if this attribute should be shown in case scenario manager project setup view.

The optional attributes do not form part of a file name, but are saved along with the file to provide additional information on the file.

The following table describes the items in **Optional attributes** groupbox.

Object	Description
Append Optional Attribute button	Append an optional attribute to the attribute list displayed in the matrix on the left.
Delete Selected Optional Attributes button	Delete the selected optional attributes in the matrix on the left.
Reorder Optional Attributes button	Manually resort the optional attributes in the matrix on the left.
Name field	The name of the attribute.
Type field	The data type of the attribute. The choices are: Text, Number, Date, Time, Yes/No.

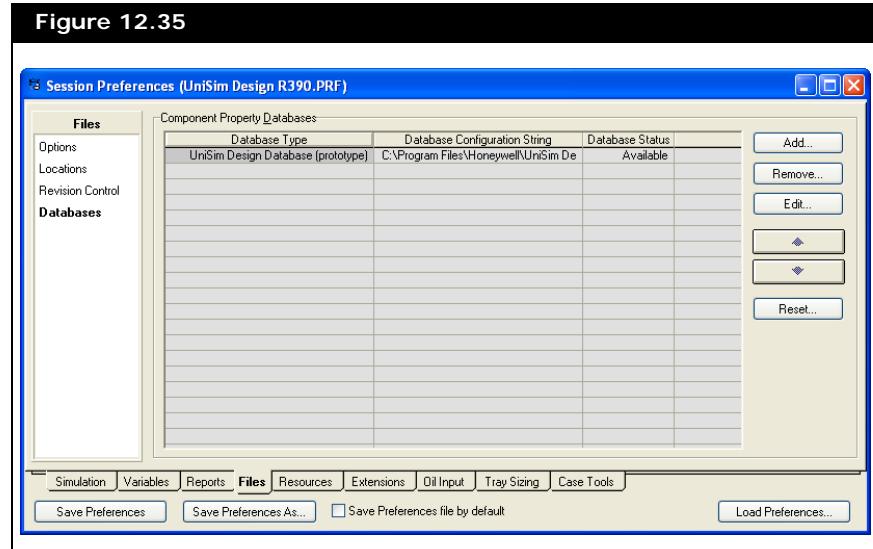
Object	Description
Format field	The format for date.
CSM Use field	Decide if this attribute should be shown in case scenario manager project setup view.

Automatically update all the date and time attributes before saving the file checkbox: Check this box if you want UniSim Design to automatically update the date and time attributes in both name creation attributes matrix and optional attributes matrix before saving the file. The corresponding fields will not be manually changeable any more.

Reset to Default button: Clicking this button will reset all the attributes to the system default.

12.5.4 Databases

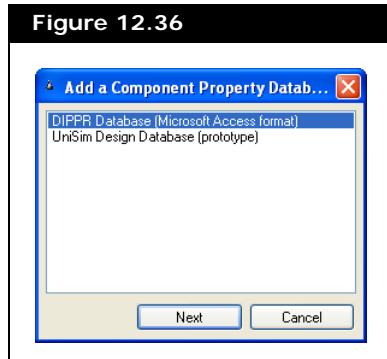
The Databases page lets you manage component property databases by adding, removing, resetting or ordering Component Property Databases. The component property database order will decide the set of available components in the component list.



To Add a Component Property Database:

1. Click the **Add** button in the Component Property Databases group.

The custom component property database type picker appears.



2. Choose the intended component property database type and Click **Next** to browse for Component Property Database file. The operation could be cancelled by clicking **Cancel**.
3. In the open file dialog, browse to the location of component property database file. From the list of available database files, select the file you want to use. Click **Open**. The selected component property database appears in the list of available component property database. You can also verify the database configuration string and database status of the added Component Property Database in Component Property Databases group.

To Remove a Component Property Database:

1. Select Component Property database to be removed from the available database list.
2. Click the **Remove** button in the Component Property Databases group.
3. UniSim Design prompt will appear to confirm the removal of the selected Component Property Database. Click **Yes** to confirm or **No** to cancel the operation.

To Edit a Component Property Database:

1. Select Component Property database from the available Component Property database list to edit the properties.
2. Click the **Edit** button in the Component Property Databases group.
3. A File Open dialog box will appear to browse and select the edited or updated component property database file.

To Change the order of Component Property Database:

1. Select Component Property Database in Component Property Databases group for which sequence needs to be changed.
2. Click the **Up Arrow** button to move the selected Component Property database higher in the order of the list.
3. Click the **Down Arrow** button to move the selected Component Property database lower in the order of the list.

Changing the order of database loader is possible only when more than one Component Property Database is configured.

Component property databases will be reset as per the last saved UniSim Session preferences

To Reset Component Property Databases:

1. Click the Reset button in the Component Property Databases group.
2. A dialog box will prompt you to confirm reset for the list of Component Property Databases. Click Yes to confirm or No to cancel the operation.

12.6 Resources Tab

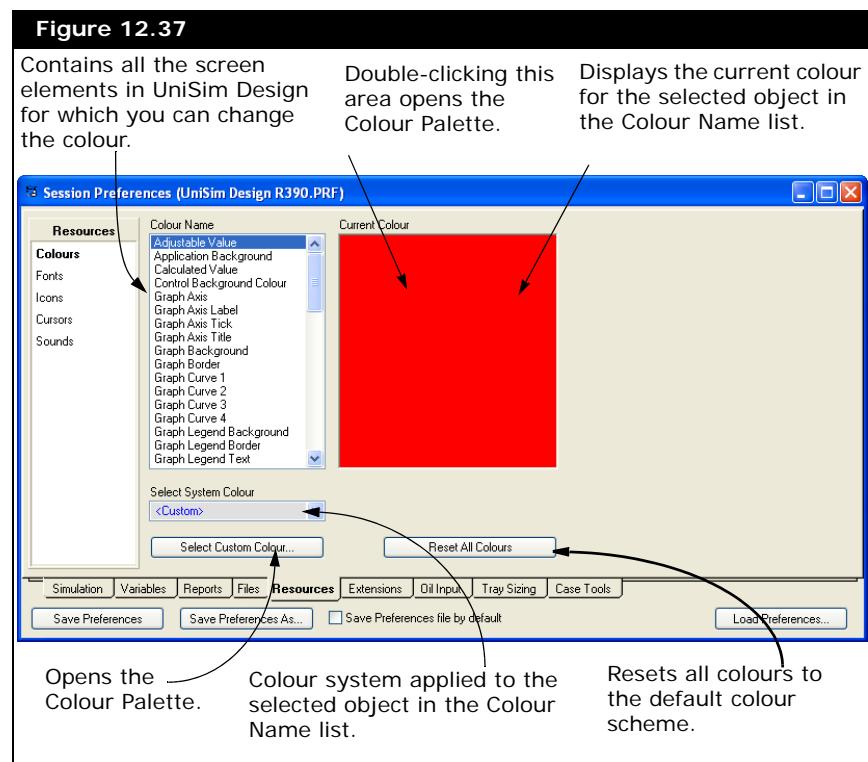
The Resources tab is divided into five pages: Colours, Fonts, Icons, Cursors, and Sounds.

12.6.1 Colours Page

UniSim Design default colour settings for text in cells/fields are as follows:

- Black text indicates the value is calculated by UniSim Design and cannot be changed.
- Blue text indicates the value is entered by the user and is editable.
- Red text indicates the value is calculated by UniSim Design and is editable.

All the functions and views in UniSim Design are set with a predefined colour scheme. The Colours page lets you customize this colour scheme to meet the specific needs of your organization or simulation.



To change the colour of an element, do the following:

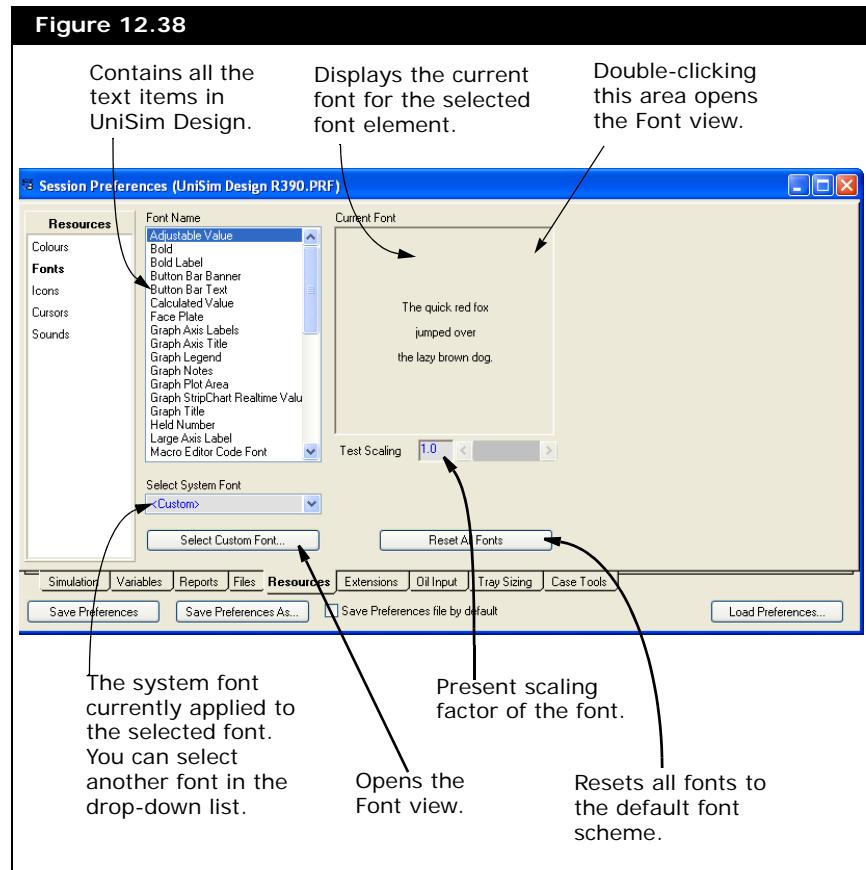
1. In the Colour Name list, select the item you want to modify.
2. From the Select System Colour drop-down list, select one of the system colours that are available, or select <Custom>.



3. If you selected <Custom>, click the **Select Custom Colour** button. The colour palette appears. Select the required colour and click **OK**.

12.6.2 Fonts Page

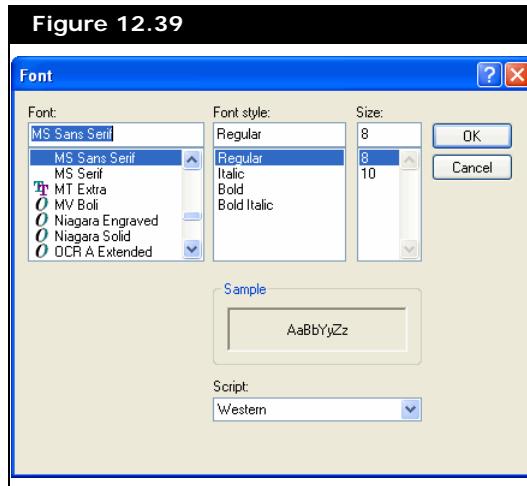
All the text displayed in UniSim Design has a predefined font scheme, however, like the colour scheme, you can change the font scheme.



To change the font of a text element, do the following:

1. In the Font Name list, select the item you want to modify.
2. From the Select System Font drop-down list, select one of the system fonts that are available, or select <Custom>.

3. If you selected <Custom>, click the **Select Custom Font** button. The Font view appears.



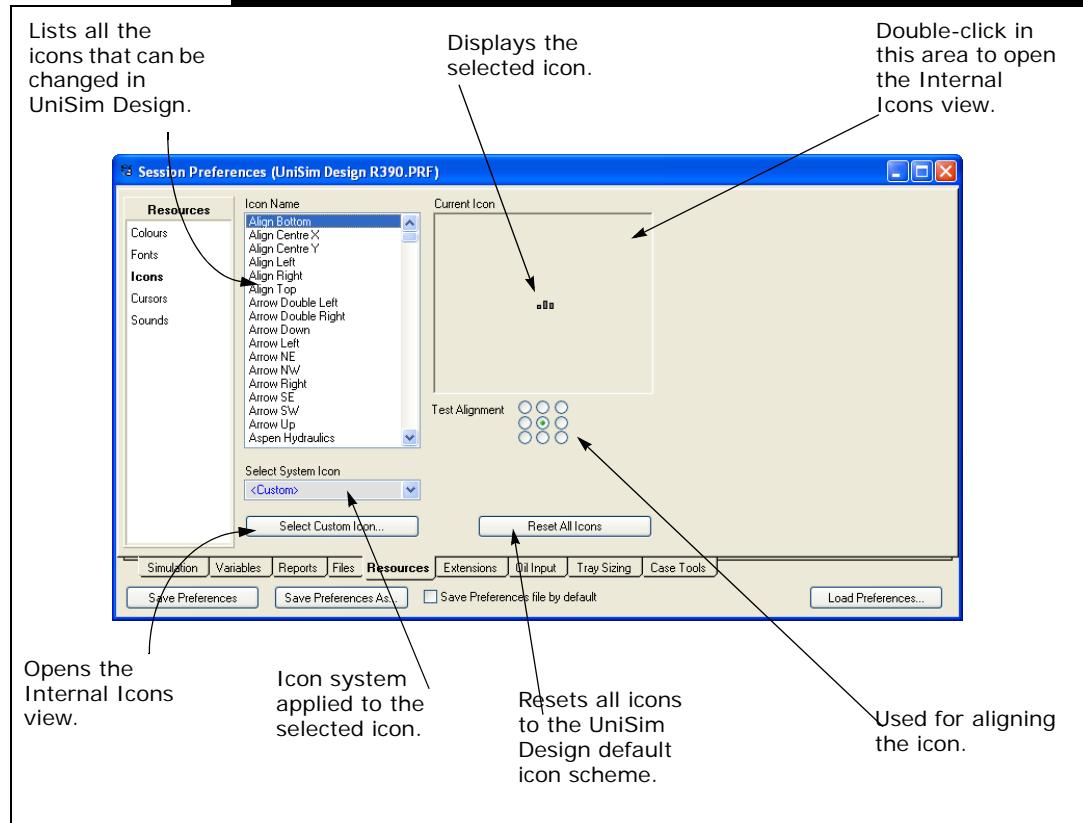
4. In the **Font** list, select the font type for the text element.
5. In the **Font Style** list, select the style for the text element.
6. In the **Size** list, select or type in the size for the text element.
7. Click **OK**.

12.6.3 Icons Page

UniSim Design has a default icon scheme. Any icons that can be

changed appear on this page.

Figure 12.40



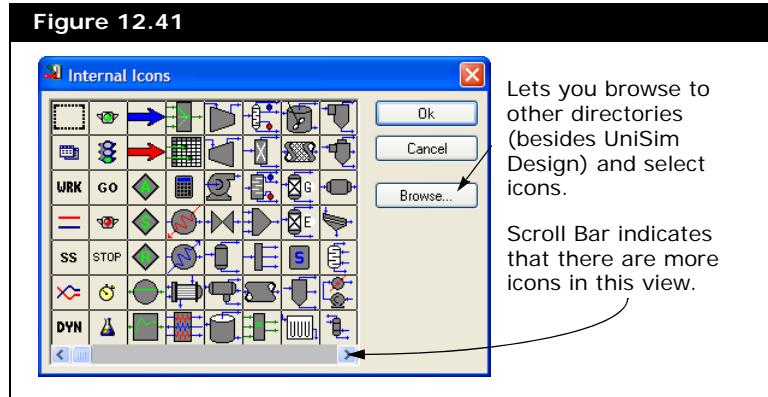
To change an icon, do the following:

1. In the Icon Name list, select the icon you want to modify.
2. From the Select System Icon drop-down list select one of the system icons that are available, or select <Custom>.

The Internal Icons view displays all the icons available in UniSim Design. You can select any of these icons as the icon for an item.

- If you selected <Custom>, click the **Select Custom Icon** button. The Internal Icons view appears. Select the required icon and click OK.

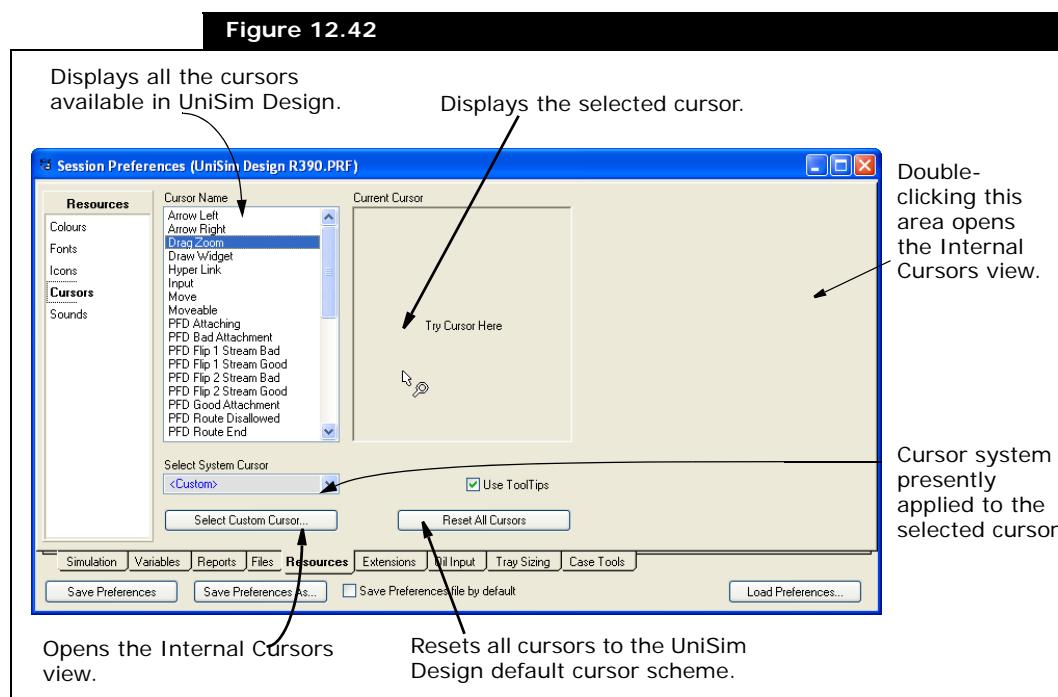
Figure 12.41



12.6.4 Cursors Page

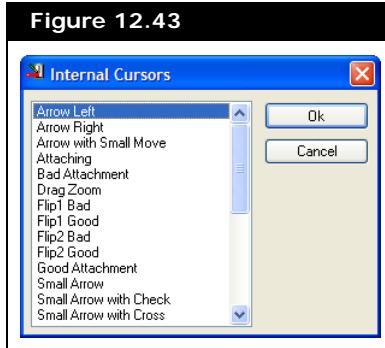
As with the Colours, Fonts, and Icons pages, you can customize the cursors in UniSim Design.

Figure 12.42



To change a cursor, do the following:

1. In the Cursor Name list, select the cursor you want to modify.
2. From the Select System Cursor drop-down list, select one of the system cursors that are available, or select <Custom>.

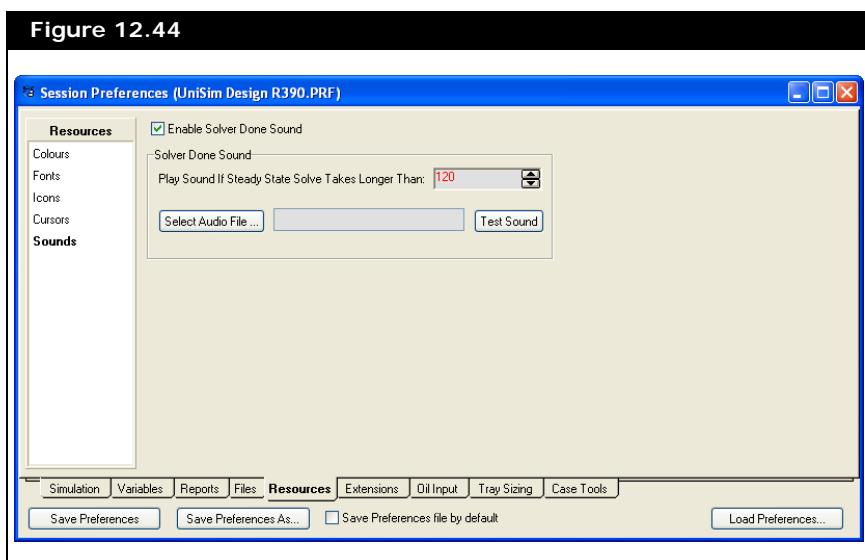


3. If you selected <Custom>, click the **Select Custom Cursor** button. The Internal Cursors view appears. Select the required cursor and click **OK**.

12.6.5 Sounds Page

Click the Test Sound button to play the selected sound file.

The Sounds page lets you set up UniSim Design to play a sound if the steady state solution takes longer than a defined period of time.



To play a sound, do the following:

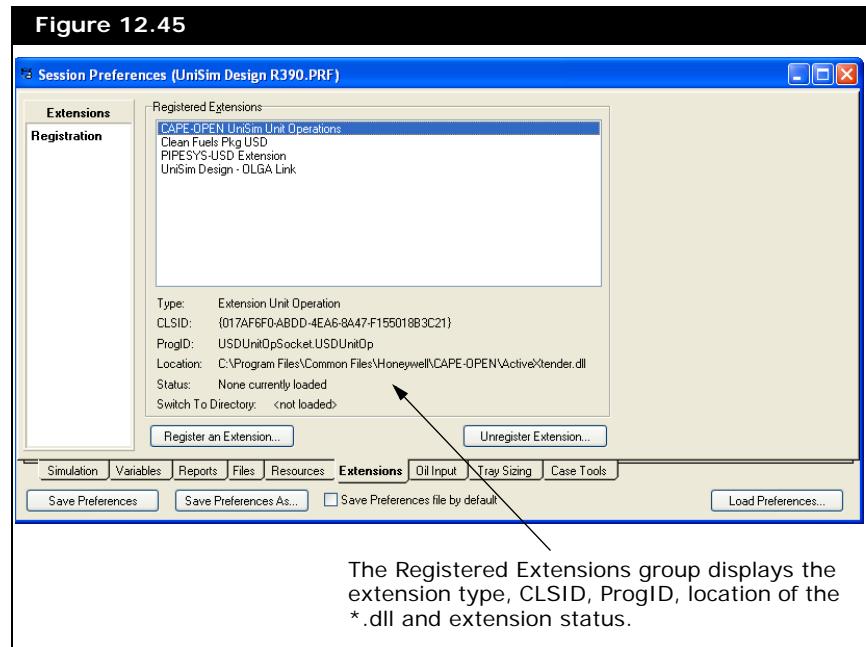
1. Check the **Enable Solver Done Sound** checkbox.
2. Click the **Select Audio File** button. The Open File view appears.
3. Browse to the location of your sound file (*.wav). Select the file you want to use, then click **Open**.

Use the up and down arrows to increase or decrease the value in the field by one each click or you can enter a value directly in the field.

- In the **Play Sound if Steady State Solve Takes Longer Than** field, enter the amount of time you want to let the steady state solver take before a sound is played.

12.7 Extensions Tab

The Extensions tab is used to register external extension in UniSim Design.



To register an extension, do the following:

- Click the **Register an Extension** button. The custom file picker appears.
- In the File Path group, browse to the location of the extension DLL.
- From the list of available extension *.dlls, select the extension you want to register with UniSim Design.
- Click **Ok**. The Custom file picker closes and the extension appears in the Registered Extensions group.

To unregister an extension, do the following:

- Select the extension you want to unregister from the list of registered extensions.
- Click the **Unregister Extension** button.

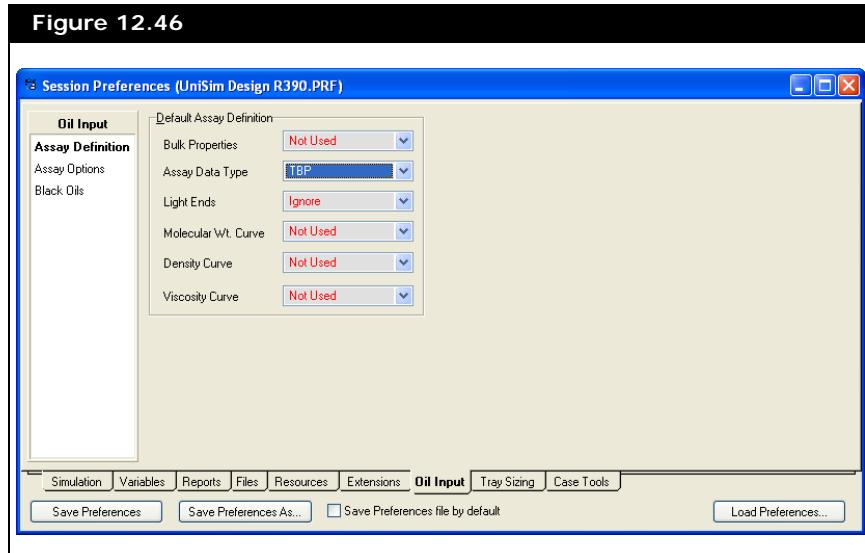
12.8 Oil Input Tab

For more details about these options, see [Chapter 3 - UniSim Design Oil Manager](#) in the [UniSim Design Simulation Basis Guide](#).

The pages on the Oil Input tab lets you set default settings for characterizing oils in your simulation case.

12.8.1 Assay Definition Page

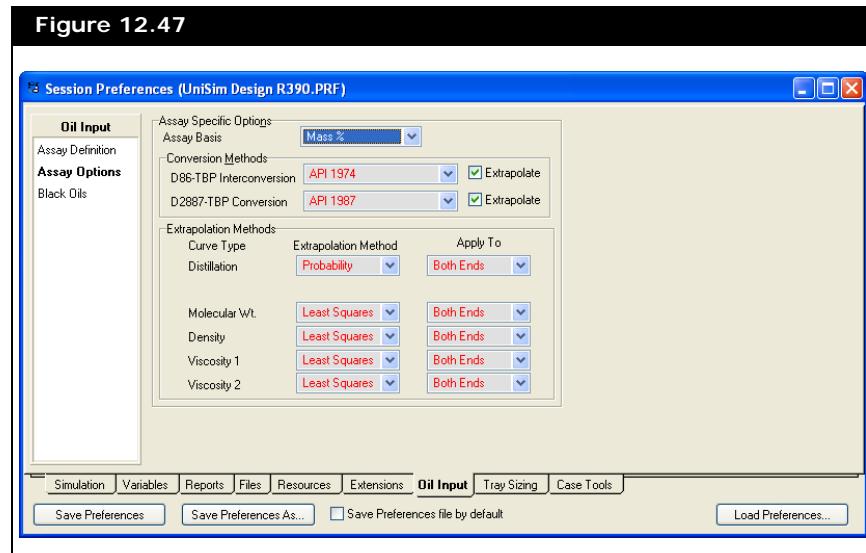
Figure 12.46



Use the following procedure to set your default assay definition.

1. From the Bulk Properties drop-down list, select Used if you are supplying bulk properties data by selection, or Not Used if you are not supplying this information.
2. From the Assay Data Type drop-down list, select one of the following data types: TBP, ASTM D86, ASTM D1160, ASTM D86-D1160, ASTM D2887, Chromatograph, EFV or None.
3. From the Light Ends drop-down list select one of the following options: Ignore, Input Composition, Or Auto Calculate.
4. From the Molecular Wt. Curve drop-down list, select one of the following options: Not Used, Dependent, or Independent.
5. From the Density Curve drop-down list, select one of the following options: Not Used, Dependent, or Independent.
6. From the Viscosity Curves drop-down list, select one of the following options: Not Used, Dependent, or Independent.

12.8.2 Assay Options Page



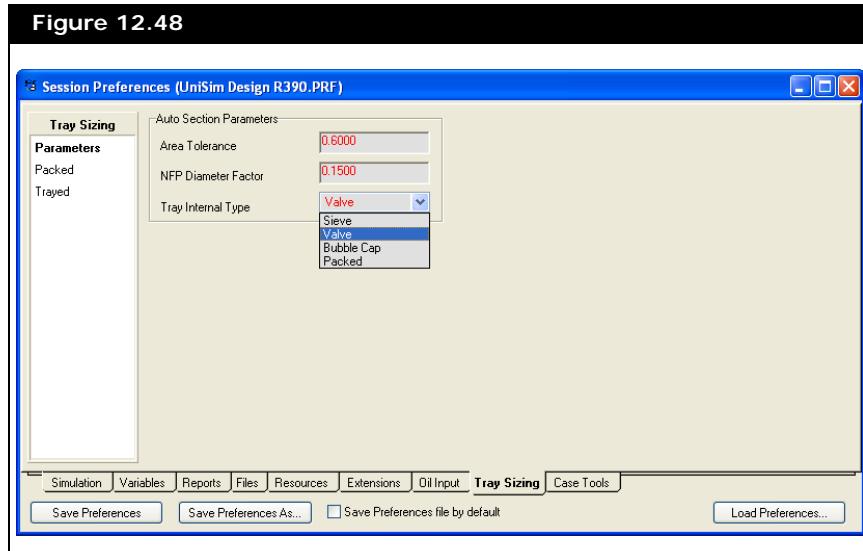
To set your default assay options, do the following:

1. From the Assay Basis drop-down list, select one of the following options: Liquid Volume%, Mass%, or Mole%.
2. Use the drop-down lists in the Conversion Methods group to select the methods UniSim Design will use for D86-TBP Interconversion and D2887-TBP Conversion.
3. Use the drop-down lists in the Curve Fitting Methods group to select the extrapolation method to be used by each curve type. There are three extrapolation methods to select from: Lagrange, Least Squares, and Probability.

12.9 Tray Sizing Tab

The Tray Sizing tab lets you set the UniSim Design defaults for the Tray Sizing utility. Any parameters set here are automatically used when you attach a Tray Sizing utility to your case. For more information about this utility, see [Section 14.18 - Tray Sizing](#) in the **UniSim Design Operations Guide**.

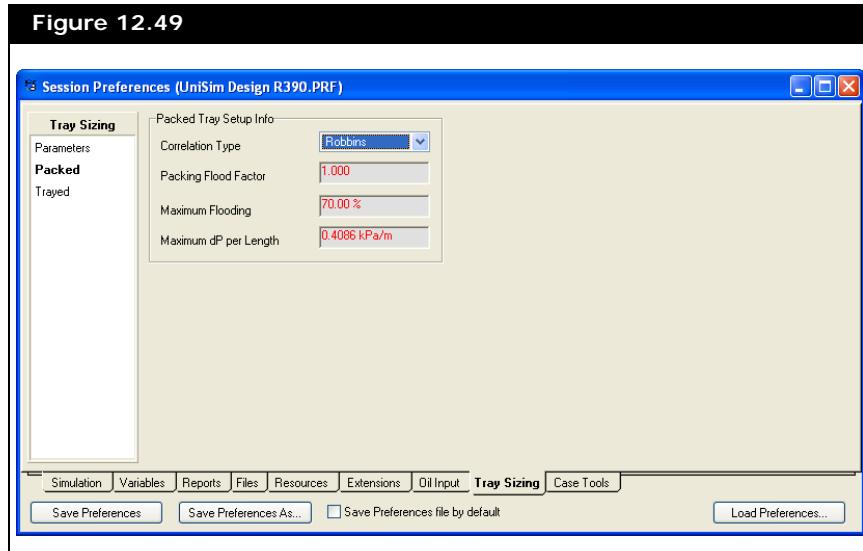
12.9.1 Parameters Page



To set your default tray sizing parameters, do the following:

1. In the **Area Tolerance** field, enter a value for your area tolerance.
2. In the **NFP Diam Factor** field, enter a value for your NFP diameter factor.
3. From the Tray Internal Type drop-down list, select one of the following tray types: Valve, Sieve, Bubble Cap, or Packed.

12.9.2 Packed Page



For information about the packing types available in UniSim Design, refer to the packinfo.db file located in the Support folder.

You can open the file using any text editor program.

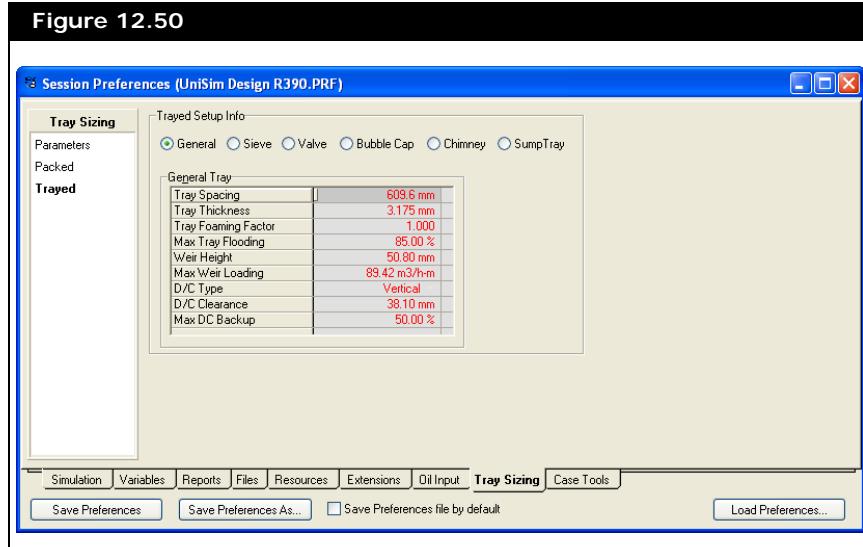
To set your default packed parameters, do the following:

1. From the Correlation Type drop-down list, select one of the following design correlations for predicting pressure drop and liquid hold-up: Robbins or Sherwood-Leva-Eckert. The default is Robbins.
2. In the **Packing Flooding Factor** field, specify the packing flood factor of your packed tray. The default is 1.000.
3. In the **Maximum Flooding** field, specify the maximum percentage of flooding you can have on your tray. The default is 70.00%.
4. In the **Maximum dP per length** field, specify the maximum pressure difference you can tolerate per measured length. The default is 0.4086 kPa/m.

12.9.3 Trayed Page

The selections made on the Trayed page are used as the default settings for every new tray section that you add to your tray sizing utility. The tray parameters group changes depending on which radio

button you select.



To set your default trayed parameters, do the following:

1. Select the **General** radio button.
2. In the General Tray group, enter values for the following tray specifications: Tray Spacing, Tray Thickness, Tray Foaming Factor, Max Tray Flooding, Weir Height, Max Weir Loading, D/C Type, D/C Clearance and Max DC Backup.
3. Select the **Sieve** radio button.
4. In the Sieve Tray group, enter values for the following sieve tray specifications: Hole Diameter, Hole Pitch, Flooding Method and Max Tray DP (height of liquid).
5. Select the **Valve** radio button.
6. In the Valve Tray group, enter values for the following valve tray specifications: Orifice Type, Design Manual, Valve Material Density, Valve Material Thickness, Hole Area (% of actual area) and Max Tray DP (height of liquid).
7. Click the **Bubble Cap** radio button.
8. In the Bubble Cap Tray group, enter values for the following bubble cap tray specifications: Hole Area (% of actual area), Cap Slot Height and Max Tray DP (height of liquid).
9. Click on the **Chimney** radio button.
10. In the Chimney Tray group, enter values for Riser Area (% of AA), Riser Height, Chimney Tray Spacing, Max Tray DP (height of liquid), Chimney Weir Height, and Residence Time.
11. Click on the **Sump Tray** radio button.
12. In the Sump Tray, enter values for Sump Tray Spacing, Liquid Height, Residence Time, and Max Tray DP (height of liquid).

Sieve tray calculations are based on the valve tray manuals for tray layout, and Mass-Transfer Operations for pressure drop, weeping and entrainment calculations by Treybal, (McGraw-Hill).

Valve tray calculations are based on the Glitsch, Koch, and Nutter valve tray design manuals.

Bubble tray calculations are based on the method described in Design of Equilibrium Stage Processes by Bufford D. Smith, (Wiley & Sons).

12.10 Case Tools Tab

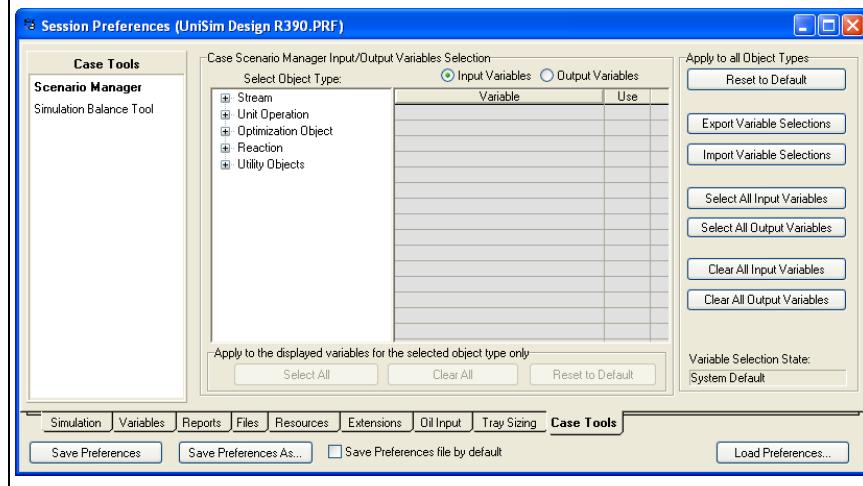
The Case Tools tab has the following pages:

- Scenario Manager
- Simulation Balance Tool

12.10.1 Scenario Manager

The Scenario Manager page lets you set the UniSim Design defaults for Case Scenario Manager's input and output variables. When a case scenario project is created, it will inherit the input and output variable settings from the preferences default. The selected input or output variables will be used to compare for differences between the master simulation case and the other cases in a case scenario project.

Figure 12.51



The following table describes the items in **Case Scenario Manager Input/Output Variables Selection** groupbox.

Object	Description
Input Variables radio button	Select the input variables to be used by the case scenario manager for case comparison. Input variables are the variables which can be modified by the user.
Output Variables radio button	Select the output variables to be used by the case scenario manager for case comparison. Output variables are the calculated variables in a simulation case.
Select Object Type object tree	Select an object type from the currently supported types of objects.

Object	Description
Variable list	Show the list of input or output variables for the selected object type.
Use checkbox	Select the input or output variables to be used in case comparison.
Select All button	Select all the displayed variables for the selected object type.
Clear All button	Unselect all the displayed variables for the selected object type.
Reset to Default button	Reset all the displayed variables to their system default selection.

The following table describes the items in **Apply to All Object Types** groupbox.

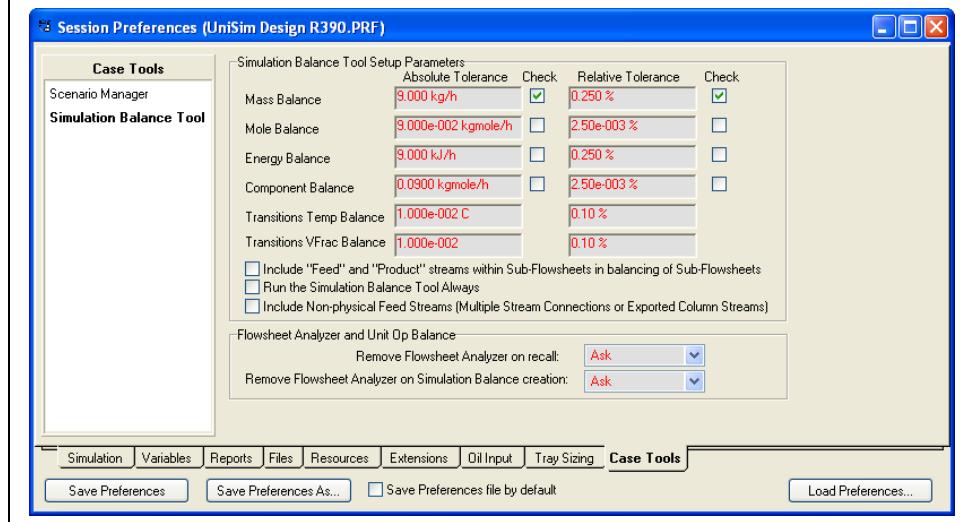
Object	Description
Reset to Default button	Reset all input and output variables to the system default variable selections for all object types.
Export Variable Selections button	Export the variable selections to a file in XML format. This file can then be imported back to the preferences or any case scenario project.
Import Variable Selections button	Import variable selections from a file in XML format. The file must be in the same format as that exported by the above export operation.
Select All Input Variables button	Select all input variables for all object types.
Select All Output Variables button	Select all output variables for all object types.
Clear All Input Variables button	Unselect all input variables for all object types.
Clear All Output Variables button	Unselect all output variables for all object types.
Variable Selection State text box	Show the current variable selection state. There are three states: <ul style="list-style-type: none"> System Default: UniSim Design's internal setting. This state is shown when user input has never been done, or when the Reset to Default button in this group has been activated. File [filename].ucsv: This state indicates that the current variable selections are imported from file [filename].ucsv, where [filename] is the name of the file without extension. Custom: This state shows that there have been some changes from above two states.

12.10.2 Simulation Balance Tool

The Simulation Balance Tool lets you set the UniSim Design defaults for the Simulation Balance Tool. Any parameters set here are automatically used when the Simulation Balance Tool is created. For more information

on the Tool itself, see [Simulation Balance Tool](#).

Figure 12.52



To set your default Tool parameters, do the following:

1. In the Mass Balance row, enter a value for your absolute tolerance, specify whether to perform the absolute mass balance check, enter a value for your relative tolerance, and specify whether to perform the relative mass balance check. The default values are 9.0 kg/h, checked, 0.25%, and checked, respectively.
2. In the Mole Balance row, enter a value for your absolute tolerance, specify whether to perform the absolute molar balance check, enter a value for your relative tolerance, and specify whether to perform the relative molar balance check. The default values are 0.09 kg-mole/h, not checked, 0.0025%, and not checked, respectively.
3. In the Energy Balance row, enter a value for your absolute tolerance, specify whether to perform the absolute heat balance check, enter a value for your relative tolerance, and specify whether to perform the relative heat balance check. The default values are 9.0 kJ/h, not checked, 0.25%, and not checked, respectively.
4. In the Component Balance row, enter a value for your absolute tolerance, specify whether to perform the absolute component molar balance check, enter a value for your relative tolerance, and specify whether to perform the relative component molar balance check. The default values are 0.09 kg-mole/h, not checked, 0.0025%, and not checked, respectively.
5. In the Transitions Temp Balance row, enter a value for your absolute tolerance and enter a value for your relative tolerance. The default values are 0.01C and 0.10%, respectively.
6. In the Transitions VFRAC Balance row, enter a value for your absolute tolerance and enter a value for your relative tolerance. The default values are 0.01 and 0.10%, respectively.

7. Select to Include "Feed" and "Product" streams within Sub-Flowsheets in balancing of Sub-Flowsheets. The default is not checked.
8. Select to Run the Simulation Balance Tool Always. The default is not checked. When this option is enabled, loading an older case without a Simulation Balance Tool will automatically create a Simulation Balance Tool.
9. Select to **Include Non-physical Feed Streams (Multiple Stream Connections or Exported Column Streams)**. The default is not checked.
10. Select to Remove Flowsheet Analyzer on recall. Selecting "Yes" will remove the feature (if exist) from the case on recall. The default is Ask.
11. Select to Remove Flowsheet Analyzer on Simulation Balance creation. Selecting "Yes" will remove the feature (if exist) from the case when the Simulation Balance Tool is created. The default is Ask.

13 Window & Help Options

13.1 Introduction	2
13.2 Window Menu.....	2
13.2.1 Save Workspace.....	2
13.2.2 Load Workspace.....	3
13.3 Help Menu.....	4
13.3.1 Adding a Bug Report.....	4
13.3.2 Editing a Bug Report.....	5
13.3.3 Deleting a Bug Report.....	5

13.1 Introduction

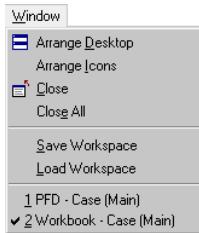
This chapter provides descriptions for the options available in the Window and Help menus.

If you want to switch focus from the menu bar without making a selection, press the **ESC** key or the **ALT** key.

To access the Window or Help menu options, do one of the following:

- Click the required menu bar item to open the associated menu.
- Use the **ALT** key in combination with the underlined letter in the menu title. For example, **ALT W** opens the **Window** menu.
- Use the **ALT** key by itself to move the active location to the **File** menu in the menu bar. When the menu bar is active, navigate across 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 to the next menu bar item, opening the associated menu.

13.2 Window Menu



This menu contains general Windows application functions. The commands are as follows:

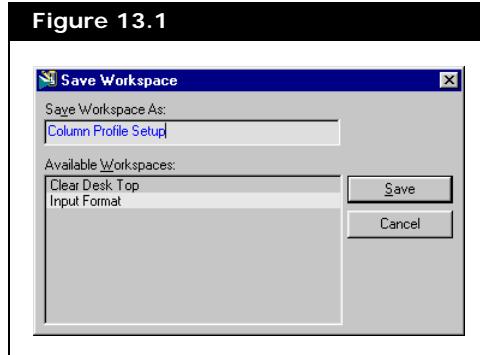
Command	Description
Arrange Desktop	Cascades all views that are currently open and not iconized. Face plates are placed in rows or column according to the specifications on the Desktop page of the Session Preferences view. Refer to Section 12.2.3 - Desktop Page , for more information.
Arrange Icons	Arranges icons horizontally at the bottom of the Desktop.
Close	Closes the active view.
Close All	Closes all views.
Save Workspace	Saves the current view layout for future use.
Load Workspace	Loads another UniSim Design case which is currently open. This function lets you toggle between cases.

The last section in this menu lists all open views on the Desktop. The active view is indicated by a checkmark.

13.2.1 Save Workspace

You can save different Workspace arrangements within a UniSim Design case. The Workspace is a specific organization of views for the current case. For example, create an arrangement of views that has the PFD, Workbook, Controllers, Strip Charts, etc. You can name each

arrangement individually, then access the arrangement at any time.



This has no effect on the calculation status. It is simply the way the various views are arranged. After changes are made to the Desktop arrangement, reload a saved arrangement to access the view layout.

1. From the **Window** menu, select **Save Workspace**. The Save Workspace view appears.
2. In the **Save Workspace As** field, enter the name of the workspace.
3. Click **Save**.

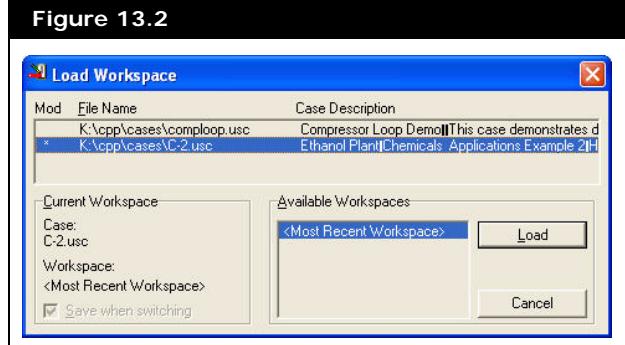
13.2.2 Load Workspace

The Load Workspace view displays all cases that are currently open and the workspaces available for the loaded case.

Loading a Workspace

Check the **Save when Switching** checkbox to save the case when switching between workspaces.

1. From the **Window** menu, select **Load Workspace**. The Load Workspace view appears.



2. In the list of available cases, select the case in which you are currently working.

3. From the list of available workspaces, select the workspace you want to use.
4. Click **Load**.

Switching to Another Open Case

1. From the **Window** menu, select **Load Workspace**. The Load Workspace view appears.
2. From the list of available cases, select the case in which you want to switch.
3. From the list of available workspaces, select the workspace you want to load in the new case.
4. Click **Load**.

Check the Save when Switching checkbox to save the case when switching between cases.

13.3 Help Menu



The following table lists and describes the commands in the Help menu:

Command	Description
UniSim Design Help Topics	Opens the UniSim Design Online Help to the Welcome page.
Help on Extending UniSim Design	Opens the UniSim Design Extensibility Online Help.
Help on the Current Form	Opens the UniSim Design Online Help to the topic that relates to the active view. If no topic is found for that view, the Welcome page appears.
Bug Reports	Accesses UniSim Design' unique bug reporting option.
About UniSim Design	Provides information about UniSim Design (version, etc.).
Honeywell on the Web	Clicking the Honeywell on the Web command opens a sub-menu with a link to the Honeywell Web site.

13.3.1 Adding a Bug Report

1. From the **Help** menu, select **Bug Reports**. The Bug Reports view appears.
2. Click the **New** button to display the Bug Report view.
3. In the **Bug Title** field, type the name of the bug report.
4. The current date and time appear in the Date field, but you can change it if necessary.

5. In the **Reported by** field, type a contact name so Honeywell Technical Support can ask further questions or provide answers to the problem.
6. In the Class group, click either the **Bug** or **Enhancement** radio button.
7. In the Priority group, click either the **High**, **Medium** or **Low** radio button.
8. In the **Program Area** field, enter the area of the program in which the bug is found.

For example, if a bug is found on the Assay tab of the Oil Characterization environment, specify Assay tab - Oil Characterization environment. Be as specific as possible so the problem can be located.
9. In the **Step-by-Step Description** field, enter the steps used to produce the bug. Be as specific as possible so the problem can be located and fixed.
10. In the Reproducibility group, click either the **Always**, **Sometimes**, or **Never** radio button.
11. Click the **Write** button to display the Save File view.
12. Specify a name and location for your bug report file.
13. Click **Save**.
14. E-mail the bug report file to the Support Center at unisim.support@honeywell.com.

13.3.2 Editing a Bug Report

1. From the **Help** menu, select **Bug Reports**. The Bug Reports view appears.
2. From the list of available bug reports, select the bug report you want edit.
3. Click the **Edit** button to display the Bug Report view.
4. Modify any of the parameters making up the bug report.

13.3.3 Deleting a Bug Report

1. From the **Help** menu, select **Bug Reports**. The Bug Reports view appears.
2. From the list of available bug reports, select the bug report you want delete.
3. Click the **Delete** button.

You are not prompted to confirm the deletion of a bug report, so ensure the correct report is selected before deleting.

Index

A

Absorber Column 8-27
Adding
 annotations in PFD 10-34
 assay 6-5
 blend 6-7
 bug report 13-4
 component list 5-4
 component map 5-22
 correlation set 6-11
 fluid package 5-11
 global correlation set 11-75
 global property correlation 11-73
 hypotheticals 5-15
 operation 8-5
 optimization object 7-62
 reaction 5-18
 reaction set 5-19
 report 9-8
 schedule/sequence/event 7-14
 stream 8-5
 sub-flowsheets 10-30
 unit set 12-21
 user password 11-60
 user property 5-24, 6-9
 user variables 7-54
 utilities 7-95
Adjust 8-32
Adjust-Recycle Manager 7-28
Air Cooler 8-10
Alarm Manager 11-85
Annotations 10-34
 adding in PFD 10-34
 editing in PFD 10-34
 moving and sizing labels 10-35
Assay 6-4
 adding 6-5
 cloning 6-6
 deleting 6-6
 editing 6-6
 exporting 6-6
 importing 6-6
Axes 10-38
Azimuth 11-36

B

Baghouse Filter 8-17
Balance 8-34
Basis Environment 5-2
 entering 7-52
Basis Menu 5-2
Boiling Ranges 6-3
Boundary Label 3-17
Bug Report 13-4
 adding 13-4

deleting 13-5
editing 13-5

C

Calculation/Responsiveness Button 1-8
Case 1-7
 snap shot 11-27
 states 11-27
Case Security 11-54
 adding user password 11-60
 changing master password 11-59
 clearing master password 11-60
 clearing user password 11-61
 file security setup 11-56
 hardware locks 11-62
 loading a lock case 11-56
 locking a case 11-54
 master password 11-59
 removing hardware locks 11-64
 scanning lock codes 11-62
 specifying lock codes 11-63
 time restriction 11-61
 unlocking a case 11-64
 user password 11-60
 viewing user password 11-60
Case Studies 11-30
 3-dimensional graph control 11-36
 adding 11-31
 multi-dimensional graphing 11-35
 removing 11-31
 setup 11-32
 viewing results 11-34
Case Summary 11-3
Cloning
 assay 6-6
 blend 6-8
 correlation set 6-13
 property correlation 11-74
 user property 6-10
Closing Commands 4-10
Colour Schemes 7-84
 setting preferences 12-36
Column
 disabling input experts 12-6
 menu 7-94
 sub-flowsheet environment 3-6
 tray section display in PFD 7-87
Column Page 12-18
Component Map 5-21
 adding 5-22
 basis environment 5-21
 deleting 5-22
 editing 5-22
Component Splitter 8-31
Components 5-3

- adding 5-4
- copying list 5-10
- deleting list 5-9
- editing list 5-4
- exporting list 5-10
- importing list 5-10
- Compressible Gas Pipe 8-12
- Compressor 8-11
- Confirm Mode Switches 12-6
- Control Manager 11-41
- Conversion Reactor 8-22
- Converting Case to Template 7-9
- Cooler 8-8
- Copy Command 10-3
- Copy Special Command 10-3
- Copy With Labels Command 10-3
- Copying**
 - component list 5-10
 - fluid package 5-13
 - reaction 5-19
 - reaction set 5-20
 - schedule/sequence/event 7-20
- Correlation Manager 11-71
 - adding global correlation set 11-75
 - adding global property 11-73
 - cloning property 11-74
 - deleting clone property 11-75
 - removing global property 11-74
- Correlation Oil Environment 6-10
- Crystallizer 8-24
- CSTR 8-19
- Cursors 12-40
- Cut Command 10-3
- CutBlend 6-6
- Cyclone 8-16
- D**
- Data Historian 11-25
 - exporting 11-26
 - See also Strip Charts*
 - viewing current data 11-26
- Data Recorder 11-27
 - deleting states 11-29
 - recording states 11-27
 - setup 11-30
- Databook 11-4
 - adding process data table 11-7
 - adding scenario 11-27
 - adding scenarios 11-27
 - adding variables 11-5
 - case studies *See Case Studies*
 - data recorder *See Data Recorder*
 - deleting process data table 11-13
 - deleting scenario 11-29
 - deleting variables 11-6
- editing variables 11-6
- process data tables 11-7
- strip charts *See also Strip Charts* 11-13
- variables 11-5
- viewing process data table 11-7
- viewing scenario 11-28
- Datasheets 12-28
 - printing 9-3
- Defining**
 - 3 phase distillation column 8-30
 - 3 phase separator 8-7
 - absorber column 8-27
 - adjust 8-32
 - air cooler 8-10
 - baghouse filter 8-17
 - balance 8-34
 - component splitter 8-31
 - compressible gas pipe 8-12
 - compressor/expander 8-11
 - conversion reactor 8-22
 - cooler/heater 8-8
 - crystallizer 8-24
 - CSTR 8-19
 - cyclone 8-16
 - digital point 8-34
 - distillation column 8-25
 - energy stream 8-6
 - equilibrium reactor 8-22
 - gibbs reactor 8-21
 - heat exchanger 8-10
 - hydrocyclone 8-16
 - liquid-liquid extractor 8-29
 - LNG exchanger 8-9
 - material stream 8-5
 - mixer 8-14
 - MPC controller 8-35
 - neutralizer 8-23
 - PFR 8-20
 - PID controller 8-33
 - pipe segment 8-13
 - precipitator 8-24
 - pump 8-11
 - reboiled absorber column 8-28
 - recycle 8-33
 - refluxed absorber column 8-26
 - relief valve 8-14
 - rotary vacuum filter 8-17
 - selector block 8-34
 - separator 8-6
 - set 8-33
 - short cut distillation column 8-31
 - simple solid separator 8-15
 - tank 8-7
 - tee 8-15
 - transfer function block 8-35

- valve 8-13
- Deleting**
 - assay 6-6
 - blend 6-8
 - bug report 13-5
 - clone property correlation 11-75
 - component list 5-9
 - component map 5-22
 - confirm mode 12-6
 - correlation set 6-12
 - event 7-20
 - fluid package 5-12
 - hypothetical group 5-16
 - optimization object 7-63
 - PFDs 7-79
 - reaction 5-19
 - reaction set 5-20
 - report 9-11
 - schedule 7-19
 - sequence 7-19
 - unit set 12-21
 - user property 5-25, 6-10
 - user variables 7-54
 - utilities 7-97
- Desktop 1-7
 - main features 1-8
 - preferences 12-10
 - toolbar 1-14
- Digital Point 8-34
- Distillation Column 8-25
- DXF Files 9-14
- Dynamic 7-50
- Dynamic Assistant 12-14
- Dynamics Assistant 7-50, 11-40
 - hot keys 1-17
 - icon 1-15
- E**
- Echo ID 11-70
- Edit Menu 10-3
- Editing**
 - assay 6-6
 - blend 6-8
 - bug report 13-5
 - component list 5-4
 - component map 5-22
 - correlation set 6-12
 - fluid package 5-12
 - hypothetical group 5-16
 - optimization object 7-62
 - PFD 10-4
 - reaction 5-19
 - reaction set 5-20
 - report 9-11
 - user property 5-25, 6-10
- F**
- Face Plates 11-39
 - opening 7-83, 11-39
 - preferences 12-10
 - types 11-40
- File Menu 4-2
 - closing commands 4-10
 - exit 4-11
 - new and open commands 4-3
 - printing 4-11
 - saving commands 4-6
- Files**
 - case 1-7
 - Custom file picker 12-30
 - hysim case 4-4
 - saving locations 12-31
 - templates 3-14, 7-9
- user variables 7-54
- Energy Stream 8-6
- Enter Basis Environment 7-52
- Environments 3-4
 - advantages 3-8
 - column sub-flowsheet 3-6
 - entering build 7-57
 - main flowsheet 3-5
 - oil characterization 3-5
 - relations 3-7
 - simulation 3-5, 7-5
 - simulation basis 3-4, 5-2
 - sub-flowsheet 3-5, 3-10
- Equation Summary 7-52
- Equilibrium Reactor 8-22
- Event Scheduler 7-13
 - adding schedule/sequence/event 7-14
 - copying schedule/sequence/event 7-20
 - deleting event 7-20
 - deleting schedule 7-19
 - deleting sequence 7-19
 - exporting schedule/sequence/event 7-23
 - importing schedule/sequence/event 7-21
 - sorting schedule/sequence/event 7-24
- Exit UniSim Design 4-11
- Expander 8-11
- Exporting**
 - assay 6-6
 - component list 5-10
 - fluid package 5-13
 - historical data 11-26
 - hypothetical group 5-16
 - reaction set 5-20
 - schedule/sequence/event 7-23
 - user variables 7-55
 - xml 7-12
- Extensions 12-42

- ufl 4-11
- xml 7-10
- Files Preferences 12-30
- Flowsheet 3-2
 - adding utilities 7-95
 - boundary label 3-17
 - column 3-3
 - deleting utilities 7-97
 - information transfer 3-13
 - multi-level architecture 3-11
 - object browser 7-58
 - special elements 1-7
 - transfer basis 3-17
 - viewing utilities 7-97
- Flowsheet Analysis 7-80
- Flowsheet Menu
 - adding operation 8-5
 - adding streams 8-5
 - boiling ranges property view 6-3
 - fluid package/dynamic model 7-63
 - notes manager 7-60
 - object navigator 7-56
 - object palette 8-3
 - optimization objects 7-61
 - reaction package 5-25, 7-63
 - simulation navigator 7-58
 - user variables *See also User Variables* 7-52
- Fluid Package 5-10
 - adding 5-11
 - associating to flowsheet 5-13
 - copying 5-13
 - deleting 5-12
 - editing 5-12
 - exporting 5-13
 - importing 5-13
 - in sub-flowsheets 3-10
- Fluid Package/Dynamic Model 7-63
- Fly-by Information Boxes 7-82
- Fonts 12-37
- Forget Pass 1-5
- Format Editor 10-41
- G**
 - Gibbs Reactor 8-21
 - Graph Control 10-36
 - 3-dimensional 11-36
 - manipulating axes 10-38
 - manipulating graph title 10-39
 - manipulating legend 10-40
 - manipulating plot area 10-40
 - modifying data display lines 10-37
 - strip charts 11-20
- H**
 - Heat Exchanger 8-10
- Heater 8-8
- Help Menu 13-4
- Help Options 13-4
- Home View Preferences 12-10
- Hot Keys 1-16
- Hydrocyclone 8-16
- Hypotheticals 5-14
 - adding 5-15
 - deleting group 5-16
 - editing group 5-16
 - exporting group 5-16
 - importing group 5-16
- HYSIM Case
 - opening in UniSim Design 4-5
 - reading 4-4
- I**
- Icons
 - changing default preferences 12-38
 - changing icons on PFD 10-18
 - toolbar 1-14
 - wire frame/3D 10-19
- Importing
 - assay 6-6
 - component list 5-10
 - fluid package 5-13
 - hypothetical group 5-16
 - reaction set 5-20
 - schedule/sequence/event 7-21
 - user variables 7-55
- Information Transfer 3-13
- Input Experts Option 12-6
- Installing
 - objects from object palette 8-3
 - oil into flowsheet 6-13
 - operation from flowsheet menu 8-5
 - operation from workbook 7-67
 - stream from flowsheet menu 8-5
 - stream from workbook 7-67
 - templates 3-20
- Integrator 7-26
 - active 7-51
 - holding 7-51
- Interface
 - basics 1-3
 - elements 1-3
 - flowsheet elements 1-7
 - terminology 1-9
- L**
 - Legacy HYSIM Case
 - limitations 4-6
 - Legend 10-40
 - License 12-17
 - Line Segments

adding bend points 10-22
 alignment 10-24
 line straightening 10-23
 moving 10-21
 removing bend points 10-23
Liquid-Liquid Extractor 8-29
LNG Exchanger 8-9
Logger Set-Up 11-14
Logger Set-Up-All 11-14
Logger Size 11-14

M

Macro Language Editor 11-52
Material Stream 8-5
Menu Bar
 access 4-2, 10-3, 11-3, 13-2
 basis 5-2
 closing commands 4-10
 column 7-94
 edit 10-3
 file 4-2
 help 13-4
 new and open commands 4-3
PFD 10-4
 printing 9-3
 saving commands 4-6
 utilities 7-94
 window 13-2
workbook *See also Workbook* 7-64
Mixer 8-14
Modal Views 12-7
Mouse Wheel Scrolling 10-20
MPC Controller 8-35
Multi-flowsheet Architecture 1-4

N

Naming Preferences 12-11
Navigation Between Flowsheets 3-13
Neutralizer 8-23
New and Open Commands 4-3
Non-Modal Views 12-7
Notes 11-25
Notes Manager 7-60

O

Object
 browser 7-58
 deselection 10-13
 moving 10-14
 multiple selection in PFD 10-12
 single selection in PFD 10-12
 transformation 10-17
 variable table 7-89
Object Navigator 7-56
 entering build 7-57

locating 7-57
 multi-flowsheet navigation 3-13
 object filter 7-57
Object Palette 8-3
Object Status Window 1-11
 object inspect menu 1-12
Oil Characterization Environment
See Environments
Oil Manager
 adding blend 6-7
 adding correlation set 6-11
 adding user property 6-9
 assay 6-4
 basis environment 5-16
 cloning blend 6-8
 cloning correlation set 6-13
 cloning user property 6-10
 correlation 6-10
 cutblend 6-6
 deleting blend 6-8
 deleting correlation set 6-12
 deleting user property 6-10
 editing blend 6-8
 editing correlation set 6-12
 editing user property 6-10
 install oil into flowsheet 6-13
 oil characterization environment 6-2
 oil output settings 6-3
 user property 6-9
Oil Output Settings 6-3, 7-55
Oils Preferences 12-43
Operations
 analysis 3-24
 connecting in PFD 10-27
 deleting from workbook 7-68
 installing from flowsheet menu 8-5
 installing from workbook 7-67
 performance 3-24
 sub-flowsheet 3-10
Optimization Objects 7-61
 adding 7-62
 deleting 7-63
 editing 7-62
Optimizer 7-12
P
Pan/Zoom Functions 10-6
See also PFD
Paste Command 10-3
Performance Page 12-16
Performance Speed 12-16
PFD 7-77
 accessing objects property views 7-80
 adding annotations 10-34
 additional functions 10-11

- aligning objects 10-15
- auto positioning 10-14
- auto snap align 10-15
- auto-scrolling 10-19
- break connection button 10-11
- break connection icon 10-30
- changing icons 10-18
- cloning objects 10-32
- closing panes 7-93
- colour schemes 12-36
- column sub-flowsheet 7-83
- column tables 7-92
- connecting logical operations 10-28
- connecting operations to streams 10-26
- connecting streams and operations 10-25
- connecting two operations 10-27
- creating streams from operation 10-26
- Cut/Copy/Paste 10-32
- Cut/Paste Functions 10-30
- deleting streams and operations 7-81
- deselecting objects 10-13
- disconnecting streams and operations 10-29
- editing 10-4
- exchanging XML files 7-94
- exporting objects 10-31
- flowsheet analysis 7-80
- hiding objects 10-35
- importing objects 10-32
- label variables 10-32
- line segments *See also Line Segments* 10-21
- modes *See also PFD Modes* 10-7
- moving objects 10-14
- multi-pane 7-92
- notebook 7-78
- object inspect menu 10-5
- object transformation 10-17
- object variable table 7-89
- open PFD option 7-82
- pan and zoom 10-6
- printing 9-5
- printing as a DFX file 9-14
- quick route 10-11
- rebuilding 10-25
- resizing panes 7-93
- reveal hidden objects 10-36
- selecting objects 10-12
- show/hide sub-flowsheet objects 7-82
- sizing objects 10-15
- stream label options 10-32
- sub-flowsheets 10-30
- swap connections 10-17
- swap connections button 10-11
- table properties 7-90
- tables 7-88
- thick stream line 10-19
- tools 10-6
- wire frame/3D icon 10-19
- working across panes 7-93
- PFD Colour Schemes 7-84
 - adding query colour scheme 7-86
 - changing 7-85
 - deleting query colour scheme 7-86
 - editing query colour scheme 7-87
 - selecting 7-85
- PFD Menu 10-4
- PFD Modes
 - attach 10-8
 - auto attach 10-8
 - move 10-8
 - size 10-8
- PFD Notebook 7-78
 - deleting a PFD 7-79
 - installing a new PFD 7-78
 - renaming a PFD 7-79
- PFR 8-20
- PID Controller 8-33
- Pipe Segment 8-13
- Playback
 - See Script Manager*
- Plots
 - control 10-36
 - printing 9-5
- Precipitator 8-24
- Preference
 - See Session Preferences*
- Printer Setup 9-6
- Printing
 - datasheets 9-3
 - menu bar 9-3
 - menu bar command 4-11
 - options 9-2
 - PFD 9-5, 10-36
 - PFD as DFX file 9-14
 - plots 9-5
 - printer setup 9-6
 - reports 9-14
 - snap shots 9-6
- Property View
 - flowsheet analysis 3-22
 - locating 7-57
- Pump 8-11
- Q**
- Quick Route 10-11
- R**
- Reaction Package 5-25, 7-63
- Reactions
 - adding 5-18
 - adding set 5-19

- adding set to fluid package 5-21
- basis environment 5-17
- copying 5-19
- copying set 5-20
- deleting 5-19
- deleting set 5-20
- editing 5-19
- editing set 5-20
- exporting set 5-20
- importing set 5-20
- Real Format Editor 10-41, 12-25
- Reboiled Absorber Column 8-28
- Recording
 - See Script Manager*
- Recycle 8-33
- Recycle-Adjust Manager
 - Simultaneous Adjust Recycle Group 7-30
- Reflux Absorber Column 8-26
- Relief Valve 8-14
- Report Builder 9-11
- Report Manager 9-7
- Report Preferences 12-26
- Reports 9-7
 - creating 9-8
 - deleting 9-11
 - editing 9-11
 - editing datasheet 9-10
 - format and layout 9-11
 - inserting datasheet 9-8
 - printing and previewing 9-14
 - removing datasheet 9-10
 - text format 9-12
- Reports, printing specs only 9-13
- Resource Preferences 12-36
- Rotary Vacuum Filter 8-17
- RTO Manager 11-90
- Runtime Mode 11-65
- S**
- Sample Interval 11-14
- Saving
 - file locations 12-31
 - save all cases 4-7-4-8
 - save as 4-7-4-8
 - saving commands 4-6
 - workspace 13-2
- Scenario Manager 12-48
- Scenarios
 - adding to databook 11-27
 - deleting 11-29
 - recording states 11-27
 - viewing in data recorder 11-28
- Script Manager 11-50
 - playback 11-52
 - recording 11-51
- Selector Block 8-34
- Separator 8-6
- Session Preferences 12-3
 - basis environment warnings 12-5
 - colour options 12-36
 - column options 12-18
 - cursor options 12-40
 - databases 12-34
 - desktop options 12-10
 - dynamics options 12-14
 - file location options 12-31
 - file saving options 12-30
 - fonts options 12-37
 - format options 12-24
 - general options 12-5
 - hypothetical components options 12-5
 - icon options 12-38
 - license options 12-17
 - load 12-4
 - modifying formats 12-25
 - naming options 12-11
 - oil assay default definition 12-43
 - oil characterizing options 12-43
 - performance setting 12-16
 - register external extension options 12-42
 - report company information options 12-29
 - report format/layout options 12-26
 - report information options 12-28
 - report text format options 12-27
 - revision control 12-32
 - save 12-3
 - saving 12-3
 - Scenario Manager 12-48
 - Simulation Balance Tool 12-49
 - sound options 12-41
 - status window options 12-18
 - steady state 12-16
 - stream property correlations 12-5
 - tool tips options 12-12
 - trace window options 12-19
 - transfer basis 12-5
 - tray sizing options 12-44
 - units options 12-20
- Set 8-33
- Short Cut Distillation Column 8-31
- Simple Solid Separator 8-15
- Simulation Balance Tool 7-98, 11-90, 12-49
- Simulation Basis Environment
 - See Environments*
- Simulation Basis Manager 5-2
- Simulation Case 1-7
 - calculation levels 7-6
 - converting case to template 7-9
 - manipulation 7-5
 - status 7-6

Simulation Environment
See Environments

Simulation Menu
 adjust-recycle manager 7-28
 dynamic/steady state 7-50
 enter basis environment 7-52
 event scheduler *See also Event Scheduler* 7-13
 importing & exporting user variables 7-54
 integrator *See also Integrator* 7-26
 main properties *See also Simulation Case* 7-5
 oil output settings 7-55
 optimizer *See also Optimizer* 7-12
 solver active/holding 7-51
 user variables *See also User Variables* 7-52
 view equations 7-52
 xml *See also XML* 7-10

Simulation Navigator 7-58
 viewing an object 7-59

Simultaneous Adjust Recycle Group 7-30

Snap Shot 11-27

Snapshot Manager 11-44

Solver Active 7-51

Solver Holding 7-51

Sorting
 schedule/sequence/event 7-24

Sounds 12-41

Spec Scenarios 11-37
 Adding 11-38
 removing 11-38
 viewing results 11-38

specs only 9-13

States
See Snap Shot

Steady State 7-50, 12-16

Step Size 7-28

Streams
 adding utilities 7-95
 allowing multiple connections 12-6
 analysis 3-22
 bend points 10-22
 defining energy stream 8-6
 defining material stream 8-5
 deleting from workbook 7-67
 deleting utilities 7-97
 installing 8-5
 installing from object palette 8-3
 installing from workbook 7-67
 manual route mode 10-21

PFD label options 10-32
 quick route mode 10-20
 view new upon creation 12-6
 viewing utilities 7-97

Strip Charts 11-13
 adding 11-15
 curves 11-16
 deleting 11-15
 graph control 11-20
 graphical line appearance 11-21
 graphical line properties 11-22–11-23
 historical data 11-25
 interval markers 11-18
 Logger Set-Up 11-14
 Logger Set-Up-All 11-14
 manipulating x-axis 11-18
 manipulating y-axis 11-17
 object inspect menu 11-20
 print control 11-25
 viewing 11-15
 zooming 11-19

Sub-flowsheets 3-10
 advantages 3-11
 capabilities 3-11
 components 3-10
 creating 10-30
 viewing from workbook 7-70

Swapping Connections in the PFD 10-11

T

Tank 8-7

Tee 8-15

Templates 3-14
 creating 3-19
 exported connections 3-15
 exported variables 3-18
 feed and product streams 3-17
 information 3-15
 installed simulation basis 3-16
 installing 3-20
 reading 3-21
 tag 3-16

Terminology Structure 1-6

Three Phase Distillation Column 8-30

Three Phase Separator 8-7

Tool Tips 12-12
 PFD 7-82

Toolbar 1-14

Tools Menu

- case security 11-54
- case summaries 11-3
- control manager 11-41
- correlation manager 11-71
- databook *See also Databook* 11-4
- dynamics assistant 11-40
- echo id 11-70
- face plates *See also Face Plates* 11-39
- macro language editor 11-52
- pfd 11-3
- reports 11-4
- script manager *See also Script Manager* 11-50
- session preferences 12-3
- utilities 11-4
- workbook 11-3
- Trace Window 1-11
 - object inspect menu 1-13
- Transfer Basis
 - flash types 3-17
 - See Flowsheet*
- Transfer Function Block 8-35
- Tray Sizing Preferences 12-44
 - packing options 12-46
 - parameters options 12-45
 - tray setting options 12-46
- U**
 - UFL Files 4-11
 - Unit Set 12-20
 - adding 12-21
 - adding conversions 12-22
 - changing 12-22
 - deleting 12-21
 - deleting conversions 12-23
 - viewing conversions 12-23
 - viewing users 12-23
 - Units Preferences 12-20
 - User Property
 - adding 5-24
 - basis environment 5-22
 - deleting 5-25
 - editing 5-25
 - oil environment 6-9
 - User Variables 7-52
 - adding 7-54
 - deleting 7-54
 - editing 7-54
 - exporting and importing 7-54
 - Utilities 7-94
 - adding 7-95
 - deleting 7-97
 - viewing 7-97
- V**
 - Valve 8-13
- Variable Navigator 11-88
 - navigator scope 11-90
 - using 11-89
- Viewing
 - user password 11-60
 - utilities 7-97
- Virtual Stream 8-36
- W**
 - Window Menu 13-2
 - load workspace 13-3
 - save workspace 13-2
 - switch workspace 13-4
 - Windows Functionality 1-3
 - Workbook 7-64
 - accessing objects 7-68
 - adding new tabs 7-72
 - deleting operations 7-68
 - deleting streams 7-67
 - deleting tabs 7-74
 - editing tabs 7-72
 - exporting 7-76
 - exporting tabs 7-76
 - importing 7-77
 - importing tabs 7-77
 - installing operations 7-67
 - installing streams 7-67
 - object hide 7-75
 - object reveal 7-75
 - object sort 7-74
 - opening 7-65
 - show name only 7-71
 - sorting information 7-74
 - tab setup 7-71
 - tables on PFD 7-88
 - variable sort 7-76
 - viewing sub-flowsheets 7-70
 - Workbook Menu 7-66
 - export 7-76
 - import 7-77
 - order hide/reveal objects 7-74
 - page scope 7-70
 - setup 7-71
 - Workspace
 - loading 13-3
 - saving 13-2
 - switching 13-4
- X**
 - XML 7-10
 - exporting 7-12
 - importing 7-12
 - printing 7-12

Z

Zoom Icons 10-6