Aspen Plus®

STEADY STATE SIMULATION

Version 10

User Guide

AspenTech®

VOLUME 3

Use of Aspen Plus and This Manual

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

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

Contents

About This Manual vii

For More Information viii

Chapter 33 **Pressure Relief Calculations** 33-1

About Pressure Relief Calculations 33-2
Creating a Pressure Relief Block 33-2
About Pressure Relief Scenarios 33-3
Selecting a Pressure Relief Scenario 33-5
Specifying the Inlet Stream for Steady State Scenarios 33-6
Specifying Initial Vessel Contents for Dynamic Scenarios 33-8
Design Rules 33-10
Specifying the Venting System 33-13
Specifying the Relief Device 33-14
Specifying the Vessel Neck 33-16
Specifying the Inlet Pipe 33-17
Specifying the Tail Pipe 33-20
Specifying Dynamic Input 33-20
Specifying Reactive Systems for Dynamic Scenarios 33-24
Specifying When to Stop Dynamic Calculations 33-25
Examining Results of Pressure Relief Calculations 33-27
Steady-State Results 33-27
Dynamic Results 33-29

Chapter 34 **Inserts** 34-1

Creating an Insert 34-2
Importing Inserts 34-2
Resolving ID Conflicts 34-3
Creating a Property Package 34-5
Using Electrolyte Inserts From the Aspen Plus Insert Library 34-6
Hiding Objects 34-7
Revealing Objects 34-7

Chapter 35 **Creating Stream Libraries** 35-1

Creating or Modifying a Stream Library 35-2
Running STRLIB Interactively 35-2
Running STRLIB in Batch Mode 35-3
STRLIB Command Summary 35-3
Description of STRLIB Commands 35-4

Chapter 36 Stream Summary Formats 36-1

About the Aspen Plus TFFs 36-2
Creating a TFF 36-3
TFF File Format and Options 36-3
TFF Sentences 36-5
Qualifier Descriptions for DISPLAY and PROP 36-9
Option Descriptions for DISPLAY and PROP 36-11
Header Sentence Order in the Stream Table 36-16
Formats for Numbers 36-16
The NORMALIZE Option 36-17
PPM, PPB, and TRACE Options 36-18

Chapter 37 Working with Other Windows Programs 37-1

About Copying, Pasting, and OLE 37-1
Copying and Pasting Simulation Data 37-2
Copying and Pasting Plots and Other Images 37-11
Creating Active Links Between Aspen Plus and Other Windows Applications 37-16
Creating Active Links Between an Aspen Plus Result and another Windows Application 37-17
Creating Active Links from a Windows Application to Aspen Plus Input Fields 37-21
Saving and Opening Files with Active Links 37-24
Saving Files with Active Links 37-24
Opening Files with Active Links 37-25
Using Embedded Objects in the Process Flowsheet Window 37-27
Embedding an Object Using Copy and Paste 37-27
Embedding an Object Using the Insert Object Dialog Box 37-27
Modifying an Embedded Object 37-28
Saving a Run With an Embedded Object 37-29

Chapter 38 Using the Aspen Plus ActiveX Automation Server 38-1

About the Automation Server 38-2
Using the Automation Server 38-2
Viewing the Properties and Methods of Aspen Plus Objects 38-3
Objects Exposed by Aspen Plus 38-3
The Aspen Plus Tree Structure 38-4
Using the Variable Explorer to Navigate the Tree Structure 38-5
Navigating the Tree Structure in the Automation Interface 38-7
Dot Notation for Navigating the Tree 38-8
Data Values 38-9
Node Attributes 38-10
Physical Quantities and Units of Measure 38-14
Retrieving Units of Measure 38-14
Converting the Units of Measure for a Value 38-15
Changing the Units of Measure for the Aspen Plus Run 38-16
Referencing Non-Scalar Variables in the Automation Interface 38-17

Accessing Variables With a Single Identifier: Column Temperature Profile 38-18
Accessing Variables with 2 Identifiers: Column Composition Profile 38-20
Accessing Variables With 3 Identifiers: Reaction Coefficients 38-22
Flowsheet Connectivity and Automation 38-25
Controlling a Simulation from an Automation Client 38-27
Members of Aspen Plus Classes 38-28
Members of Class IHApp 38-28
Members of Class IHNode 38-30
Members of Class IHNodeCol 38-32
Members of Class IHAPEngine 38-34

Chapter 39 **Heat Exchanger Design Program Interface** 39-1

About the Heat Exchanger Design Program Interface 39-2
Generating Property Data in a Simulation 39-2
Starting HTXINT 39-3
Selecting Heating/Cooling Curve Results to Export 39-5
Generating the Interface File 39-7
Using the Interface File in Your Heat Exchanger Design Program 39-8

Index Index-1

About This Manual

The *Aspen Plus User Guide* consists of three volumes that provide step-by-step instructions for using Aspen Plus® to build and use a process simulation model.

Volume 1 describes the Aspen Plus user interface and explains how to perform the basic tasks for creating and running simulations. Topics include:

- Creating a simulation model
- Defining the flowsheet
- Entering the required information, such as components, streams and physical property data
- Running the simulation
- Examining results

Volume 2 contains procedures for using additional Aspen Plus capabilities:

- Convergence
- Sensitivity
- Design specifications
- Optimization
- Property analysis
- Data regression

Volume 3 contains information about:

- Pressure relief calculations
- Stream libraries
- Working with other Windows™ programs
- The Aspen Plus ActiveX® automation interface

For More Information

Online Help Aspen Plus has a complete system of online help and context-sensitive prompts. The help system contains both context-sensitive help and reference information. For more information about using Aspen Plus help, see the *Aspen Plus User Guide*, Chapter 3.

Aspen Plus Getting Started Building and Running a Process Model This tutorial includes several hands-on sessions to familiarize you with Aspen Plus. The guide takes you step-by-step to learn the full power and scope of Aspen Plus.

Aspen Plus Getting Started Modeling Processes with Electrolytes This tutorial includes several hands-on sessions to familiarize you with simulating electrolyte systems with Aspen Plus.

Aspen Plus Getting Started Modeling Petroleum Processes This tutorial includes several hands-on sessions to familiarize you with simulating petroleum processes with Aspen Plus.

Aspen Plus Getting Started Customizing Unit Operation Models This tutorial includes several hands-on sessions to familiarize you with the customization of unit operation models with Aspen Plus.

Aspen Plus User Guide The three-volume *Aspen Plus User Guide* provides step-by-step procedures for developing and using an Aspen Plus process simulation model. The guide is task-oriented to help you accomplish the engineering work you need to do, using the powerful capabilities of Aspen Plus.

Aspen Plus reference manual series Aspen Plus reference manuals provide detailed technical reference information. These manuals include background information about the unit operation models and the physical properties methods and models available in Aspen Plus, tables of Aspen Plus databank parameters, group contribution method functional groups, and a wide range of other reference information. The set comprises:

- *Unit Operation Models*
- *Physical Property Methods and Models*
- *Physical Property Data*
- *User Models*
- *System Management*
- *System Administration*
- *Summary File Toolkit*

Aspen Plus application examples A suite of sample online Aspen Plus simulations illustrating specific processes is delivered with Aspen Plus.

Aspen Plus Installation Guides These guides provide instructions on platform and network installation of Aspen Plus. The set comprises:

- *Aspen Plus Installation Guide* for Windows
- *Aspen Plus Installation Guide* for OpenVMS
- *Aspen Plus Installation Guide* for UNIX

The Aspen Plus manuals are delivered in Adobe portable document format (PDF) on the Aspen Plus Documentation CD.

Technical Support

World Wide Web For additional information about AspenTech products and services, check the AspenTech World Wide Web home page on the Internet at:

http://www.aspentech.com/

Technical resources To obtain in-depth technical support information on the Internet, visit the Technical Support homepage. Register at:

http://www.aspentech.com/ts/

Approximately three days after registering, you will receive a confirmation e-mail and you will then be able to access this information.

The most current Hotline contact information is listed. Other information includes:

- Frequently asked questions
- Product training courses
- Technical tips

AspenTech Hotline If you need help from an AspenTech Customer Support engineer, contact our Hotline for any of the following locations:

If you are located in:	Phone Number	Fax Number	E-Mail Address
North America & the Caribbean	+1-617/949-1021 +1-888/996-7001 (toll free)	+1-617/949-1724	support@aspentech.com
South America			
(Argentina office)	+54-11/4393-5308	+54-11/4394-8621	tecnoba@aspentech.com
(Brazil office)	+55-11/5506-0756	+55-11/5506-0567	tecnosp@aspentech.com
Europe, Gulf Region, & Africa			atesupport@aspentech.com
(Brussels office)	+32-2/724-0100	+32-2/705-4034	
(UK office)	+44-1223/312220	+44-1223/366980	
Japan	+81-3/3262-1743	+81-3/3262-1744	atjsupport@aspentech.com
Asia & Australia			atasupport@aspentech.com
(Hong Kong office)	+85-2/2838-6077	+85-2/2833-5642	
(Korea office)	+82-2/761-5800	+82-2/761-5803	

33 Pressure Relief Calculations

This chapter describes how to use the Pressure Relief (Pres-Relief) features of Aspen Plus to:

- Determine the steady-state flow rating of pressure relief systems
- Dynamically model vessels undergoing pressure relief due to a fire or heat input specified by the user.

Topics in this chapter include:

- About pressure relief calculations
- Setting-up pressure relief calculations
- Specifying information for the scenarios
- Specifying the composition and conditions
- Defining the venting system
- Specifying chemical reactions
- Specifying when to stop the simulation (stop criteria)
- Specifying the over pressure factor and piping pressure drop rules
- Examining calculation results

About Pressure Relief Calculations

Use Pressure Relief to simulate a vessel undergoing pressure relief or for simple valve rating. Pressure Relief uses the same physical property models and data as other Aspen Plus flowsheet models.

The modeling equations for nozzle flow, and for bubbly and churn-turbulent disengagement are based on technology developed by the Design Institute for Emergency Relief System (DIERS) Users Group of the AIChE. This technology is considered the best available for pressure relief system design. The Aspen Plus Pipeline model simulates flow through the inlet and tail pipes in the relief system.

Pressure Relief always operates in rating mode. This means that the program will calculate the pressure profile in the vessel and piping, given the size of the relief device. In addition, you must specify the:

- Dimensions of the equipment being protected and a connecting nozzle if present
- Pressure relief scenario
- Dimensions of inlet and tail piping, if present
- Dimensions of the relief device

Each Pressure Relief block models one scenario and one vessel. To model more than one scenario or pressurized vessel in an Aspen Plus run, include more than one Pressure Relief block in the simulation. Pressure Relief blocks are not part of the simulation flowsheet (there is no icon needed), but they can reference simulation streams.

Pressure Relief analyzes the specified scenario and reports:

- Rated capacity
- Results profiles (temperature, pressure, vapor fraction)
- Whether the system meets design rules that you select or that applicable codes (such as ASME) require

Creating a Pressure Relief Block

To create a Pressure Relief block:

1. From the Data menu, point to Flowsheeting Options, then Pres-Relief.
2. In the Pressure Relief Object Manager, click New.
3. In the Create New ID dialog box, enter an ID (name) or accept the default ID.

4. Click OK.

   The Pressure Relief Setup form appears.

See the remaining sections of this chapter for information on completing the Pressure Relief specifications.

About Pressure Relief Scenarios

A Pressure Relief scenario is a situation that causes venting to occur through the relief system. There are four types of generic scenarios to choose from:

- Steady-state flow rating of relief system (valve and piping)
- Steady-state flow rating of relief valve (no piping)
- Dynamic run with vessel engulfed by fire
- Dynamic run with specified heat flux into vessel

Each scenario is described briefly below.

Steady-State Flow Rating of Relief System Scenario

Use this scenario to find the flow rate through an emergency relief system, given the condition of the stream flowing into it and the upstream and downstream pressures. The relief system may include a relief valve a vessel neck, and two segments each, of inlet and tail pipes, as well as any number of block valves and fittings. In this scenario the pressure relief model calculates the steady-state flow rate through the specified system.

Steady-State Flow Rating of Relief Valve Scenario

Use the valve rating scenario when:

- You know the pressure, temperature, and stream composition
- You want to find out the valve capacity

The differences between the Valve rating scenario and the Relief System Rating scenario are that in the Valve Rating scenario:

- No piping is allowed
- The relief device must be a Process Safety Valve (PSV) or a Process Safety Valve-Process Safety Disk (PSV-PSD)

Dynamic Run With Vessel Engulfed by Fire Scenario

Pressure Relief provides three standards for computing the fire exposure scenario:

- NFPA 30
- API 520
- API 2000

The chosen fire standard determines which regulations are used to calculate fire scenario factors such as vessel wetted area, energy input, and credit factors.

Pressure Relief assumes the calculated energy input is constant during the entire venting transient. If appropriate, you can specify individual credit factors for drainage, water spray, and insulation to reduce the energy input. Alternatively, you can specify an overall credit factor.

Dynamic Run With Specified Heat Flux Into Vessel Scenario

The heat input scenario is similar to the fire exposure scenario, except:

- You choose the energy input value
- Credit factors are not allowed

There is no cut-off time for the duration of the event. The specified heat flux can be constant, from a constant temperature source, or a function of time.

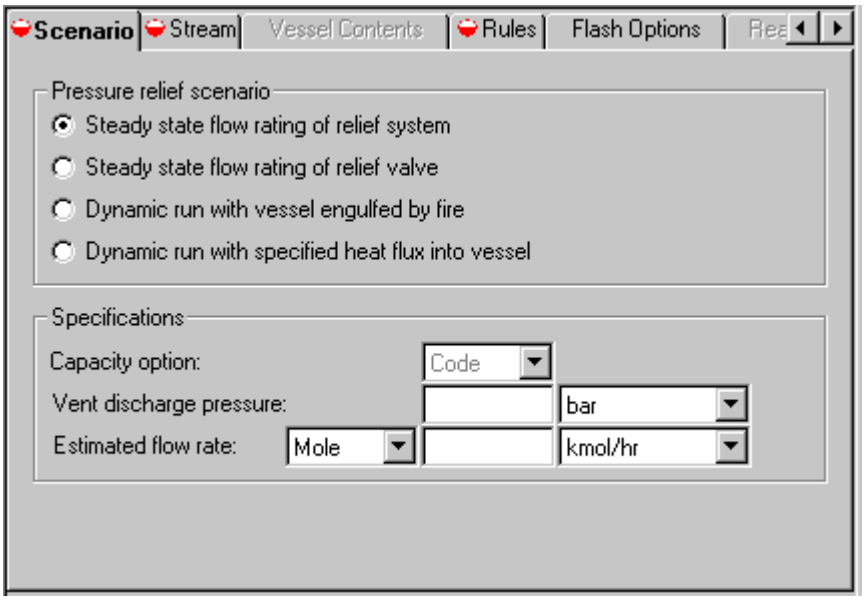
This scenario can be used to model:

- Full-on electrical heaters or other constant energy sources, by selecting the Constant Duty heat input method.
- Pressure relief caused by runaway reactions, by using the Constant Duty heat input method and specifying a value of zero for heat duty.
- Heat input from a source such as heat exchanged by selecting the Calculated from the Heat Source method and providing a source temperature, a heat transfer coefficient and surface area.

Selecting a Pressure Relief Scenario

To specify a pressure relief scenario:

1. On the Pressure Relief Setup Scenario sheet, click one of the four choices of pressure relief scenario.



2. From the Capacity option list, select Code or Actual.

Choose this option	To run the simulation at	Meaning	This option is useful for
Code (default)	Code capacity	De-rates capacity of relief device as specified by ASME code requirements.	Determining if a pressure relief valve or rupture disk of a given size is adequate for the chosen scenario.
Actual	Actual capacity	Produces best estimate of relief system effluent. Does not de-rate capacity.	Checking whether the inlet or tail pipe sections meet the code compliant requirements.

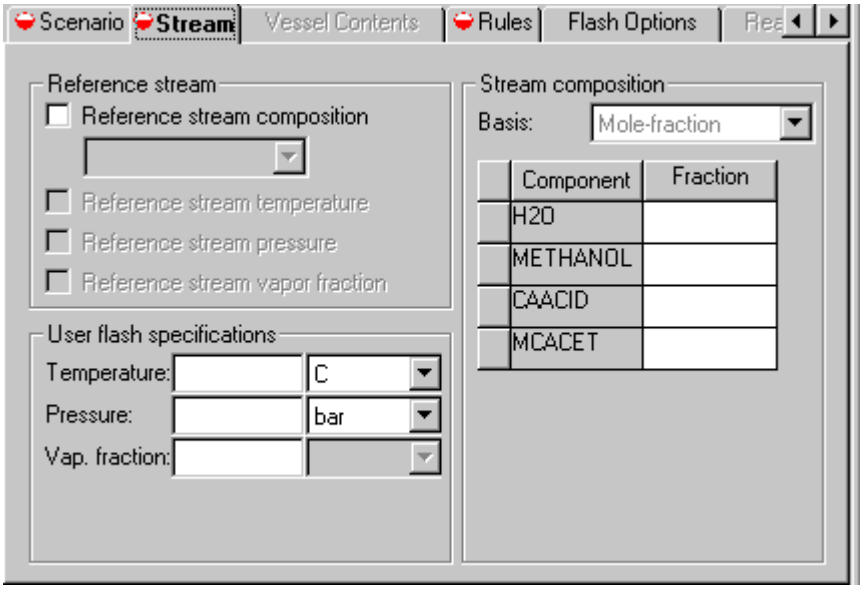
3. Specify the Vent Discharge Pressure. You need to specify this for all pressure relief scenarios. If there are any tail pipe segments after the relief device the discharge pressure refers to the pressure at the end of piping. This is typically the atmospheric pressure of the flare header back pressure.
4. If you selected a steady state flow scenario, enter the Estimated Flow Rate.
5. The value you enter is used as a starting point to determine the rated flow for the relief system or safety valve. You can enter this flow in a mass, mole or standard liquid volume basis.

The scenario you choose on the Setup Scenario sheet determines which of the remaining Pressure Relief forms and sheets you need to complete to define your system. The required forms will be displayed as incomplete in the left pane of the Data Browser Forms and sheets that do not apply to your chosen scenario are inactive.

Use Next to guide you through the required forms.

Specifying the Inlet Stream for Steady State Scenarios

For the steady state scenarios of relief valve or relief system rating you must provide the composition of the stream entering the relief valve or system.



You can:

- Specify the composition of a stream or vessel by referencing a stream from the flowsheet

  – or –

- Specify the stream composition directly in the pressure relief block

These specifications are entered on the Stream sheet of the Pressure Relief Setup form.

To specify the composition of a stream or vessel by referencing a stream from the flowsheet:

1. Select the Stream tab from the Pressure Relief Setup form.

2. On the Stream sheet, click the Reference Stream Composition check box.
3. In the field below the check box, select the desired stream.

To specify the stream composition directly in the pressure relief block:

1. Select the Stream tab from the Pressure Relief Setup form, to open that sheet.
2. In the Stream Composition frame of the Stream sheet, choose a composition basis from the Basis list.
3. Enter the component fractions in the Fraction fields next to each component present in the stream.

In addition to the composition, you must specify the thermodynamic conditions of the inlet stream. This involves specifying some combination of the three state variables: temperature, pressure, and molar vapor fraction. These values can either be entered directly on the Stream sheet, or referenced from the flowsheet if you have referenced a stream composition.

To reference an inlet stream state variable from a flowsheet stream:

1. On the Stream sheet, first ensure that you have checked the Reference Stream Composition check box. You cannot reference state variables from a stream if you have not also referenced the composition.
2. In the Reference Stream frame of the Stream sheet, click the check box indicating which stream variable you wish to reference:
   - Reference Stream Temperature
   - Reference Stream Pressure
   - Reference Stream Vapor Fraction

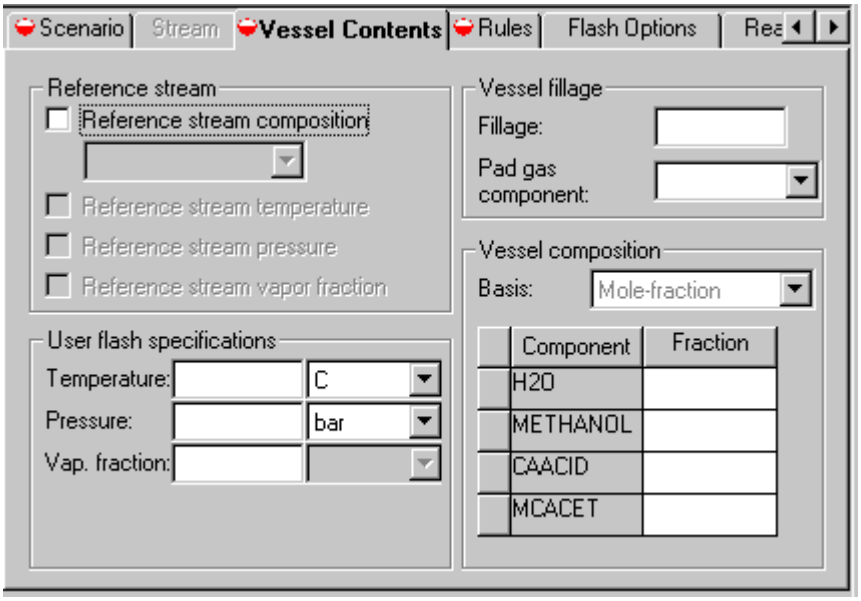
To specify a state variable for the inlet stream directly in the pressure relief block:

1. On the Stream sheet, enter a value in one of the fields of the User Flash Specifications frame:
   - Temperature
   - Pressure
   - Vap. Fraction
2. Select the appropriate units for the entered value.

   If you wish, you can reference one state variable, and enter the other directly.

Specifying Initial Vessel Contents for Dynamic Scenarios

For the dynamic scenarios of fire or heat input, you must provide the initial conditions of the vessel on the Vessel Contents sheet of the Pressure Relief Setup form.



The initial conditions of the vessel can be specified in terms of:

- Composition
- Thermodynamic conditions (temperature, pressure or vapor fraction)
- Fillage
- Pad gas component

The composition of the vessel must be specified for all pressure relief blocks, however the remaining specification can be specified using one of the following three combinations:

- Two of temperature, pressure, and molar vapor fraction
- Two of temperature, pressure, and fillage
- All of temperature, pressure, fillage, and pad gas component

The pressure relief model uses this information and the volume you define on the Pressure Relief ReliefDevice form to calculate the initial mass in the system.

Specifying Vessel Composition for Dynamic Scenarios

The vessel composition can be provided by either:

- Referencing a flowsheet stream composition
- Specifying values directly on the Vessel Contents sheet

To specify the composition of the vessel by referencing a stream from the flowsheet:

1. Select the Vessel Contents tab from the Pressure Relief Setup form.
2. On the Vessel Contents sheet, click the Reference Stream Composition check box.
3. In the field below the check box, select the desired stream ID.

To specify the stream composition directly in the pressure relief block:

1. Select the Vessel Contents tab from the Pressure Relief Setup form.
2. In the Vessel Composition frame of the Vessel Contents sheet, choose a composition basis from the Basis list.
3. Enter the component fractions in the Fraction fields next to each component present in the vessel.

Defining Thermodynamic Conditions for Dynamic Scenarios

To define the thermodynamic conditions, you must specify some combination of the state variables temperature, pressure, or vapor fraction. The required combination depends on whether you specify fillage and pad gas component.

When specifying temperature, pressure, and molar vapor fraction, you can either:

- Enter a value directly on the Vessel Contents sheet
- Reference a flowsheet stream from which to retrieve the value

To reference a vessel state variable from a flowsheet stream:

1. On the Vessel Contents sheet, first ensure that you have checked the Reference Stream Composition check box. You cannot reference state variables from a stream if you have not also referenced the composition.
2. In the Reference Stream frame of the Vessel Contents sheet, click the check box indicating which stream variable you wish to reference:
   - Reference Stream Temperature
   - Reference Stream Pressure
   - Reference Stream Vapor Fraction

To specify a state variable for the vessel directly on the Vessel Contents sheet:

1. On the Vessel Contents sheet, enter a value in one of the fields of the User Flash Specifications frame:
   - Temperature
   - Pressure
   - Vap. Fraction
2. Select the appropriate units for the entered value.

If you wish, you can reference one state variable, and enter another directly.

Fillage

Fillage is the initial fraction of the vessel volume filled with liquid (liquid holdup). The fillage must be greater than zero and less than 0.994.

If you specify the fillage but not a pad-gas component, the temperature or pressure specification will be used to determine the bubble point (vapor pressure for pure components) of the initial mass.

To specify fillage:

➤ On the Vessel Contents sheet, enter a value in the Fillage field of the Vessel Fillage frame.

Pad Gas Component

Pad Gas Component represents the component being added to bring the pressure up to the specified level. For example, nitrogen is often used as a pad gas in hydrocarbon storage tanks.

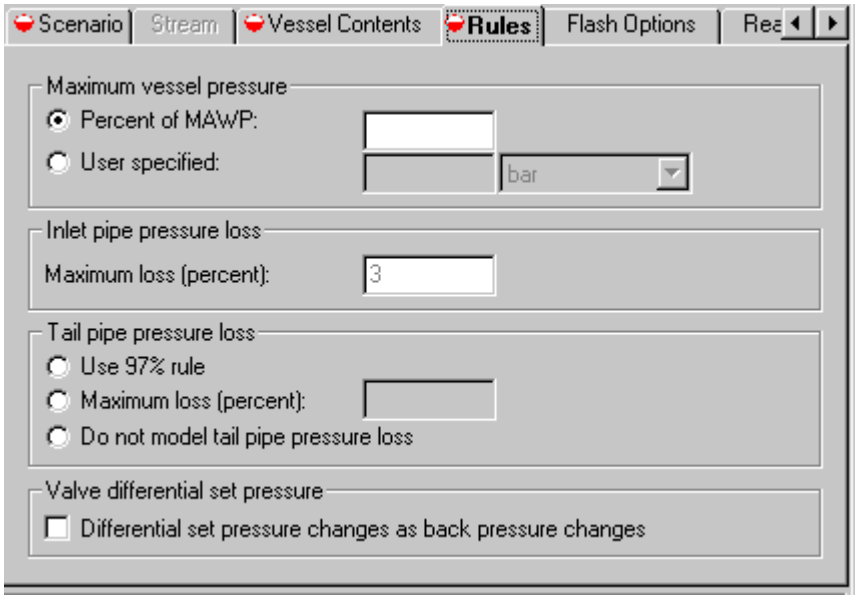
To specify a pad gas component:

➤ On the Vessel Contents sheet, choose a component from the Pad Gas Component list in the Vessel Fillage frame.

Design Rules

Use the Rules sheet of the Pressure Relief Setup form to specify rules regarding:

- Maximum vessel pressure (dynamic scenarios only)
- Inlet pipe pressure loss
- Tail pipe pressure loss
- Valve differential set pressure



Except for the limit on maximum vessel pressure, these rules apply only when the relief device is selected to be a PSV (safety relief valve) or PSV in combination with a PSD (rupture disk) for gas or two-phase service.

These rules have been included as an aid for good design practice. Pressure Relief will generate warnings if any of the rules are violated. However, any design or safety analysis decision involving these rules should be based on your own interpretation of the relevant codes and design practices.

Each section of the Rules sheet is summarized below. See *Aspen Plus Unit Operation Models* for a detailed discussion of design rules.

The following table summarizes the application of the design rules:

Rule	Application	Device
Inlet pressure loss	At 10% over-pressure†	Gas/2 phase service PSV
Tail pressure loss	At 10% over-pressure††	Gas/2 phase service PSV
97% rule	At or above 10% over-pressure†	Gas/2 phase service PSV
Max vessel pressure	Always	All devices

† *If 10% over-pressure is not reached, the highest pressures are scaled to 10% over-pressure. If all pressures are above 10% over-pressure, these rules are not applied and a warning is issued.*

Maximum Vessel Pressure

For the dynamic scenarios of fire or heat input, you must provide the maximum vessel pressure. This value can be expressed as an absolute value in the User Specified field, or as a percentage of the maximum allowable working pressure (MAWP) entered on the Design Parameters sheet of the Pressure Relief DynamicInput form.

When using the Percent of MAWP specification, MAWP is converted to gauge pressure before Aspen Plus applies the percentage entered.

Inlet Pipe Pressure Loss

Use this field to enter the maximum inlet piping pressure loss as a percentage of differential set pressure. This is calculated at 10% over-pressure or maximum pressure if 10% over-pressure is not reached. Aspen Plus generates a warning if the total pressure loss in the inlet piping is greater than or equal to the specified percentage of differential set pressure. The specified value is usually 3 (the default), and this rule is often called the “3% rule.”

Tail Pipe Pressure Loss

Use this frame for specifying the method for setting the allowable tail pipe pressure loss. You can do one of the following:

- Use the 97% Rule
- Enter the maximum loss expressed as a percentage of differential set pressure (known as the X% Rule)
- Specify that Pressure Relief not model tail pipe pressure loss

When the 97% rule is used, Aspen Plus generates a warning if the valve pressure drop is less than 97% of the valve's differential set pressure. When the X% rule is used, Aspen Plus generates a warning if the pressure loss after the valve is equal to or greater than X% of the valve differential set pressure.

The following tables suggests which rule should be used for the most common types of safety valves:

Valve Type	Suggested Tail Piping Rule
Standard spring loaded	97% Rule or X% with X=10
Pop action pilot with unbalanced pilot vented to discharge	97% Rule or X% with X=10
Balanced bellows spring loaded	X% with X=30
Modulating pilot with balanced pilots or pilots vented to atmosphere	X% with X=40

Valve Differential Set Pressure

You can specify whether the differential set pressure (DSP), changes when back pressure changes (that is, whether the valve is balanced or vented).

The following tables indicates whether the DSP changes for the most common types of safety valves:

Valve Type	Does DSP change?
Standard spring loaded	Yes
Pop action pilot with unbalanced pilot vented to discharge	Yes
Balanced bellows spring loaded	No
Modulating pilot with balanced pilots or pilots vented to atmosphere	No

Specifying the Venting System

A Pressure Relief venting system can consist of the following components:

- A vessel neck (nozzle)
- Up to two lengths (segments) of inlet pipe connecting the vessel neck to the pressure relief device
- The pressure relief device (safety valve, rupture disk, relief vent, or combination rupture disk and safety valve)
- Up to two lengths (segments) of tail pipe from the relief device to the atmosphere or to another piece of equipment

You do not have to include all these components in a Pressure Relief calculation. But the block must include at least a pipe section or a relief device.

Use the Configuration sheet on the ReliefDevice form to specify:

- Relief device type, including service for safety relief valves
- Number of inlet and tail pipe sections
- Whether the vessel neck is to be specified
- Whether the vessel neck and piping should be ignored during dynamic runs

If you choose Open Vent Pipe as the device, then:

- No relief device is allowed
- You must specify a vessel neck, inlet pipe, or tail pipe

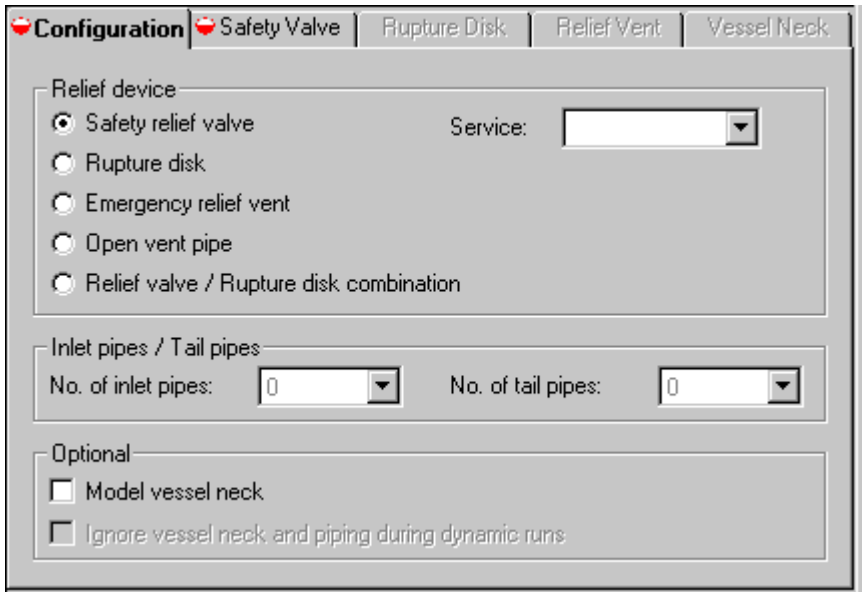
For the Steady State Flow Rating of Relief Valve scenario, the venting system consists only of a safety valve. No piping is allowed.

Specifying the Relief Device

The following types of relief devices are available:

- Safety relief valve (both liquid and gas/2-phase)
- Rupture disk
- Emergency relief vent
- Open vent pipe (pipe section or vessel neck)
- Relief valve / Rupture disk combination

On the Configuration Sheet of the ReliefDevice form, choose the type of relief device you wish to simulate by clicking on one of the options above. The default relief device is a safety relief valve.



Depending on the type of relief device you choose on the Configuration sheet of the ReliefDevice form, one or more additional sheets (tabs) on the ReliefDevice form may become active for further specification of the device. For example, if you select Rupture disk as your relief device on the Configuration sheet, the Rupture Disk tab will become active and display an incomplete symbol indicating that further specifications are required on this sheet.

Each of these additional sheets is explained later in this section.

Built-in tables within Aspen Plus contain:

- Several standard commercially available valves, rupture disks, and emergency relief vents
- All the mechanical specifications and certified coefficients needed in the relief calculations

You can customize Aspen Plus by modifying or adding tables of valve, disk, and vent characteristics. For more information, see *Aspen Plus System Management.*

You can do one of the following:

- Choose a device from the tables
- Enter your own specifications and coefficients

For liquid service valves, you can specify the full-lift over-pressure factor. This allows you to simulate some of the older-style valves, which do not achieve full lift until 25% over-pressure is reached.

Safety Valve

If you select Safety Relief Valve or Relief Valve / Rupture Disk Combination as the type of relief device, you must complete the Safety Valve sheet to specify the safety relief valve (PSV) to be used in the simulation. Define the valve in the Manufacturer's Tables frame. Once the Type, Manufacturer, Series, and Nominal Diameter have been selected, a unique valve is described and Aspen Plus fills in the following data in the Valve Parameters frame:

- Inlet Diameter
- Throat Diameter
- Outlet Diameter
- Discharge Coefficient

If you want to use a valve not listed in the Manufacturer's Tables, you must type in values for the Valve Parameters listed above. Note that if you select a valve from the tables and then overwrite any of the Valve Parameters, all Manufacturer's Tables fields will be blanked out.

To complete the form, enter the differential setpoint for the valve. This represents the pressure difference across the valve which is needed for the valve to start opening.

Rupture Disk

If you select Rupture Disk or Relief Valve / Rupture Disk Combination as the type of relief device, you must complete the Rupture Disk sheet to specify the rupture disk (PSD) to be used in the simulation. Define the rupture disk in the Manufacturer's Tables frame. Once the Manufacturer, Style, and Nominal Diameter have been selected, a unique PSD is described and Aspen Plus fills in the following data in the Rupture Disk Parameters frame:

- Diameter
- Discharge Coefficient

If you want to use a PSD not listed in the Manufacturer's Tables, you must type in values for the parameters listed above. Note that if you select a rupture disk from the tables and then overwrite any of the rupture disk parameters, all Manufacturer's Tables fields will be blanked out.

To complete the form, enter the differential setpoint for the rupture disk. This represents the pressure difference across the rupture disk which is needed for the disk to break.

In an actual capacity run, the rupture disk is modeled as a pipe with an equivalent length to diameter (L/D) ratio. If no test data is available, use L/D=8 for disk diameters larger than 2 inches (5.08 cm), and 15 for diameters 2 inches and smaller. In a code capacity run, the rupture disk is modeled as an ideal nozzle with the appropriate discharge coefficient. For uncertified rupture disks, use a discharge coefficient of 0.62.

Relief Vent

If you select Emergency Relief Vent as the type of relief device, you must complete the Relief Vent sheet to specify the emergency relief vent (ERV) to be used in the simulation. Define the relief vent in the Manufacturer's Tables frame. Once the Manufacturer, Style, and Nominal Diameter have been selected, a unique ERV is described and Aspen Plus fills in the following data in:

- Recommended Setpoint
- Diameter

To complete the form, enter the differential setpoint for the relief vent. This represents the pressure difference across the vent which is needed for the vent to begin opening.

If you want to use an ERV not listed in the Manufacturer's Tables, you must type in values for the diameter and differential setpoint in the Vent Parameters frame. Note that if you select an ERV from the tables and then overwrite any of the vent parameters, all Manufacturer's Tables fields will be blanked out.

ERVs are modeled so that they open gradually to a fully open position calculated using the vent over-pressure factor.

Specifying the Vessel Neck

The vessel neck is a piece of pipe which connects the vessel to the first length of inlet pipe or to the relief device if there are no inlet pipes. If you choose to specify a vessel neck, enter the associated information on the Vessel Neck sheet of the ReliefDevice form.

The following data are required to describe the vessel neck:

- Length
- Diameter

Optional specifications on the Vessel Neck sheet are shown below with their corresponding defaults:

Specification	Default
Orientation	Vertical
Connection Type	Rounded
Reducer resistance coefficient (Reducer K)	0.04
Expander resistance coefficient (Expander K)	0.04
Roughness	0.00015 ft

Specifying the Inlet Pipe

For inlet pipe sections, use the Pressure Relief InletPipes form to describe the inlet pipes which connect the vessel neck to the relief device. Up to two sections of pipe with the same or different diameters may be used.

The InletPipes form contains four sheets:

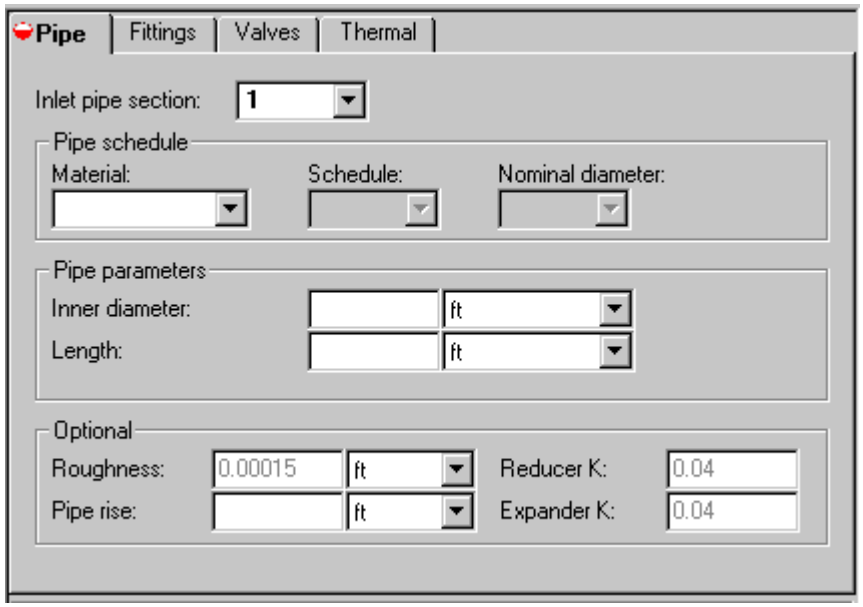
Use this sheet	To
Pipe	Specify the pipe dimensions and optional pipe parameters
Fittings	Describe the fittings in the pipe section such as pipe connections, butterfly or gate valves, elbows, and tees
Valves	Describe a general purpose valve or control valve in the pipe section
Thermal	Specify heat transfer parameters for energy transfer with surroundings

The Pipe sheet is required for all sections of inlet pipe.

The remaining sheets are used for specification of optional input. Each sheet is described below.

Pipe

Use the Pipe sheet to enter the pipe diameter and length for each pipe section. Pipe diameter and length are required specifications for all pipe sections.



Included with Aspen Plus are built-in Pipe Schedule Tables that aid in the specification of common pipe sizes.

You can customize Aspen Plus by adding or modifying pipe schedule tables. For more information on this, see the *Aspen Plus System Management* manual.

To choose a pipe diameter from the built-in Pipe Schedule Tables:

1. At the top of the Pipe sheet, select a pipe section from the Pipe Section field.
2. In the Pipe Schedule frame, choose a material of construction for the pipe section, from the Material list. Available materials include carbon steel and stainless steel.
3. Choose a pipe schedule in the Schedule field.
4. Choose a nominal pipe diameter from the Nominal Diameter field.

A unique pipe is described and the Inner Diameter is displayed in the Pipe Parameters frame.

If you want to use a pipe not listed in Pipe Schedule Tables, you must manually enter the pipe inner diameter in the Inner Diameter field of the Pipe Parameters frame. Note that if you select a pipe from the tables and then overwrite the inner diameter, all Pipe Schedule Tables fields will be blanked out.

Optional inputs on the Pipe sheet include:

- Absolute pipe roughness
- Pipe Rise (elevation change)
- Resistance coefficient ("K" factor) of reducer following the pipe section
- Resistance coefficient ("K" factor) of expander following the pipe section

If you do not enter values for these optional inputs, the following default values are used.

Specification	Default
Roughness	0.00015 ft
Pipe Rise	0
Reducer K	0.04
Expander K	0.04

Fittings

Use the Fittings sheet to describe any fittings contained in the an inlet pipe section. Any number of the following fitting types may be specified:

- Gate Valves
- Butterfly Valves
- 90 degree Elbows
- Straight Tees
- Branched Tees

In addition a miscellaneous flow resistance may be specified by entering in the number of pipe diameters equivalent to the resistance.

Valves

Use the Valves sheet to specify a general purpose valve to be used in a pipe section. In the top section of the sheet, select a valve from the Manufacturer's Tables frame. Once the Manufacturer, Style, and Nominal Diameter have been selected, a unique valve is described and Aspen Plus fills in the following data in the Valve Parameters frame:

- Flow Area
- Flow Coefficient

If you want to use a valve not listed in the Manufacturer's Tables, you must type in values for the parameters listed above. Note that if you select a valve from the tables and then overwrite any of the valve parameters, all Manufacturer's Tables fields will be blanked out.

You may also specify the valve constant for a control valve contained in the pipe section. To do this, enter a value in the Valve Constant field of the Control Valve frame.

Thermal

Use the Thermal sheet to specify energy balance parameters if you want to model heat transfer between the pipe section and the surroundings. The following must be specified:

- Inlet Ambient Temperature
- Outlet Ambient Temperature
- U-value (overall heat transfer coefficient)

Specifying the Tail Pipe

For tail pipe sections, use the Pressure Relief TailPipes form to describe the tail pipes which connect the relief device to the discharge point. Up to two sections of pipe with the same or different diameters may be used. The TailPipes form has the same functionality as the InletPipes form, except that it applies to tail pipe sections. For more details on specifying tail pipes, refer to Specifying the Inlet Pipe, earlier in this chapter.

Specifying Dynamic Input

Use the DynamicInput form to describe the emergency event associated with a dynamic scenario. Required data include a description of the vessel as well as the event (fire or heat input) which is causing the release from the vessel.

The sheets contained on the DynamicInput form are listed below:

Use this sheet	To specify
Vessel	Vessel and vessel head geometry
Design Parameters	Vessel design pressure and disengagement model
Fire	Type of fire exposure engulfing the vessel
Fire Credits	Credits you can claim if systems to fight fire or minimize vessel releases are present
Heat Input	The rate of heat input into a vessel

The Vessel and Design Parameters sheets must be completed for all dynamic scenarios. For heat input scenarios, the Heat Input sheet is also required. For fire scenarios, the Fire sheet is required, and the Fire Credits sheet is optional.

The use of each sheet is described below.

Vessel

Use this sheet to describe the vessel undergoing the emergency event by specifying:

- Vessel type and head type
- Shell orientation if the vessel type is Heat Exchanger Shell
- Vessel length and diameter
- Vessel jacket volume if vessel type is Vessel Jacket
- Vessel volume if vessel is User Specified
- Head volume and head area if head type is User Specified

The following vessel types are available:

- Horizontal
- Vertical
- API Tank
- Sphere
- Heat Exchanger Shell
- Vessel Jacket
- User Specified

To complete vessel specifications:

1. On the Pressure Relief DynamicInput Vessel sheet, choose a vessel type from the Vessel Type list in the Vessel Description frame.
2. For vessel types of Horizontal, Vertical, API Tank, or Heat Exchanger Shell, select a head type in the Head Type list, and enter the length and diameter in the Vessel Dimensions frame. Available options for head type are Flanged, Ellipsoidal, or User Specified.
3. For Heat Exchanger Shell vessel types, enter the shell orientation (horizontal or vertical) in the Shell Orientation list.
4. For Sphere vessel types, enter the sphere diameter in the Diameter field of the Vessel Dimensions frame.
5. For Vessel Jacket vessel types, enter the jacket volume in the Vessel Dimension frame.
6. For User Specified vessel types, enter the vessel volume and head volume in the User Specifications frame.
7. If you choose a User Specified head type, enter the head volume and head area in the User Specifications frame.

Design Parameters

Use the Design Parameters sheet to describe vessel design characteristics by specifying:

- Maximum allowable working pressure (MAWP)
- Temperature corresponding to the MAWP
- Vessel disengagement model
- Homogeneous vapor fraction limit for user specified disengagement model
- Volume of vessel internals

Vessel disengagement models let you select how the phase behavior of fluid leaving the vessel will be modeled. The following disengagement options are available:

Option	Description
Homogeneous	Vapor fraction leaving vessel is the same as vapor fraction in vessel
All vapor	All vapor leaving vessel
All liquid	All liquid leaving vessel
Bubbly	DIERS bubbly model
Churn-turbulent	DIERS churn-turbulent model
User specified	Homogeneous venting until vessel vapor fraction reaches the user-specified value, then all vapor venting

To specify design parameters for your dynamic pressure relief scenario:

1. On the Pressure Relief DynamicInput Design Parameters sheet, first enter the Maximum Allowable Working Pressure in the Vessel Design Pressure frame.
2. In the MAWP Temperature field, enter the temperature corresponding to the specified maximum allowable working pressure.
3. In the Vessel Disengagement frame, select a disengagement model from the Disengagement Model list.
4. Enter any vessel disengagement parameters corresponding to the selected disengagement model:

If you choose	Enter
Bubbly	Bubbly disengagement coefficient (default is 1.01)
Churn-turbulent	Churn-Turbulent disengagement coefficient (default is 1)
User-specified	Homogeneous vapor fraction limit (no default)

Optionally, at the bottom of the sheet, you can enter the Volume of Vessel Internals in the Vessel Dead Volume frame. This represents the volume of internal structures such as mixers and baffles that will be subtracted from the calculated vessel volume.

Fire

Use this sheet to characterize the fire engulfing the vessel. A Fire Standard must be selected so that Aspen Plus can calculate the vessel wetted area, energy input, and how the credit factor (Fire Credits sheet) is taken. Pressure Relief can base its calculations on any of the following standards:

- NFPA 30
- API 520
- API 2000

Choose the desired standard from the Fire Standard list in the Fire Scenario Parameters frame.

Optional specifications on the Fire sheet include:

- Fire duration
- Area of vessel surrounded by fire (relevant when Vessel Jacket or User Specified is selected for vessel type on the Vessel sheet)
- Liquid level in the vessel at the start of the fire (for NFPA-30 fire standard with Vessel Types of Horizontal, Vertical, API-Tank, or Heat Exchanger Shell)
- Vessel elevation
- Extra heat transfer area
- Whether vessel is portable

The details of how wetted areas and energy inputs are calculated are described in on-line help and in *Aspen Plus Unit Operation Models*.

Fire Credits

Pressure Relief assumes the calculated energy input is constant during the entire venting transient. If appropriate, you can use the Fire Credits sheet to specify a fire credit factor which Aspen Plus uses to reduce the energy input into the vessel. You may specify a credit factor directly or allow Aspen Plus to calculate the credit factor based on the presence of the following systems:

- Water spray equipment
- Drainage system
- Vessel insulation
- Both drainage and fire fighting equipment

Credit for Drainage and Fire Fighting Equipment is not allowed unless Fire Standard (Fire sheet) is API-520.

Insulation Protection Factor is not allowed when vessel is portable (Fire sheet), Fire Standard (Fire sheet) is NFPA-30, or Credit for insulated vessel (Fire Credits sheet) is not claimed.

The details of how credit factors are calculated are described in on-line help and in *Aspen Plus Unit Operation Models.*

Heat Input

Use this sheet to specify the rate of heat input into the vessel, for dynamic heat input scenarios. The method of heat input can be specified in three ways by selecting one of the following options in the Heat Input Method frame on the Heat Input sheet.

- Constant duty
- Calculated from heat source
- Time-varying duty profile

If a constant duty is chosen, simply enter the constant duty in the Heat Input Method frame.

If you choose to calculate the duty from a heat source, you must specify the temperature, heat transfer area, and heat transfer coefficient (U-value) in the Heat Source frame.

If you choose to enter a time-varying duty profile, use the Duty Profile frame to enter values of duty versus time.

Specifying Reactive Systems for Dynamic Scenarios

For the dynamic scenarios of fire or heat input, you can model a protected vessel with reactions if the Vessel Type (DynamicInput Vessel sheet) is one of the following:

- Vertical
- Horizontal
- API-Tank
- Sphere
- User-specified

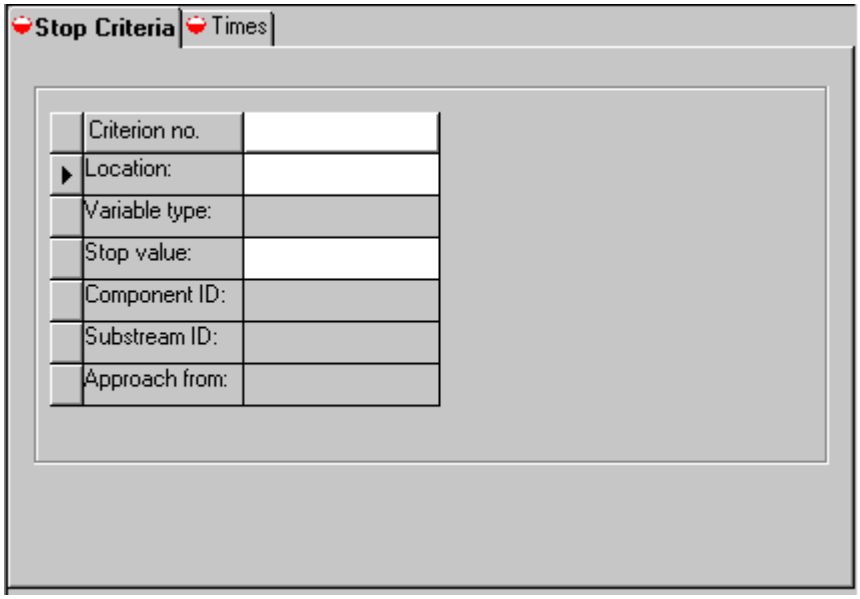
In the Reaction sheet of the Setup folder, specify which reactions occur within the vessel. If there are any kinetic or equilibrium (other than electrolytic) type reactions, you must create a Reaction ID through the Reactions folder of the Data Browser. Electrolytic reactions specified through the Chemistry folder are automatically included in the pressure relief calculations and need not be specified in the Reactions sheet. See Chapter 27 for more information on how to specify a Reaction ID.

To specify reactions to be used in a pressure relief calculation:

1. Make sure to select one of the dynamic scenarios (fire or heat input) on the Setup Scenario sheet.
2. From the Pressure Relief Setup form, click the Reactions tab.
3. On the Reactions sheet, click the Include Chemical Reactions in Vessel check box.
4. In the Vessel Reactions frame, select the desired reaction ID from the Available list, and move them to the Selected list using the right arrow button. To remove reactions from the Selected list, select them and click the left arrow button. Use the double arrow buttons to move all reactions in a selected list.

Specifying When to Stop Dynamic Calculations

When simulating the dynamic scenarios of fire or heat input, you must use the Pressure Relief Operations form to describe the criteria that Aspen Plus will use to terminate the dynamic simulation.



On the Stop Criteria sheet of the Operations form, define one or more stop criteria. You must define at least one stop criterion for the form to be complete. If you define more than one stop criteria, the criterion reached first will end the simulation.

You may select from the following variable types when defining a stop criterion:

- Simulation time
- Vapor fraction in the vessel
- Mole fraction of a specified component
- Mass fraction of a specified component
- Conversion of a specified component
- Total moles or moles of a specified component
- Total mass or mass of a specified component
- Vessel temperature
- Vessel pressure
- Vent mole flow rate or mole flow rate of a component
- Vent mass flow rate or mass flow rate of a component

You must:

- Select a specification type
- Enter a value for the specification at which the simulation will stop
- Select a component and substream for component-related specification types
- Specify which approach direction (above or below) to use in stopping the simulation.

1. To specify when to stop calculations for dynamic pressure relief scenarios:
2. Open the Pressure Relief Operations form.
3. On the Stop Criteria sheet, enter 1 in the Criterion No. field if this is the first stop criterion. When entering multiple stop criteria, number them sequentially, starting with 1.
4. From the Location list, choose a location (Vessel, Vent, or Vent Accumulator) for calculating the variable that will be defined as the stop criterion.
5. In the Variable Type list, choose a stop criterion variable type from the list shown above.
6. In the Stop Value field, enter the value of the variable at which to stop the simulation.
7. For component-related specification types, select a component and a substream from the Component ID list and the Substream ID list respectively.
8. In the Approach From list, specify which approach direction (above or below) to use in stopping the simulation.
9. Repeat steps 2 through 7 for each additional stop criterion.
10. When finished defining all stop criteria, click the Times tab.
11. On the Times sheet, specify an upper limit for the time of the scenario to be simulated, in the Maximum Time field.

12. In the Time Intervals Between Result Points frame, enter the time interval to report result for the scenario. Enter this value in the When Vent is Closed field. Aspen Plus will report result profiles in the time interval you specify. If you wish to use a different interval when the vent is open, enter this value in the When Vent is Open field. If you do not enter a value for When Vent is Open, the interval specified for When Vent is Closed will be used throughout the entire scenario.
13. In the Optional frame at the bottom of the sheet, you can request to include profile points when the relief system opens or closes. This option is checked by default. Click this option to deselect it.
14. Also in the Optional frame, you can limit the maximum number of result points in the profile by entering a value in the Maximum No. of Result Points field. The default for this value is the specified maximum time divided by the report time interval for results you have specified for When Vent is Closed.

    If you have specified a results time interval for When Vent is Open that is smaller than When Vent is Closed, you should increase the default value for Maximum No. of Results Points.

Examining Results of Pressure Relief Calculations

To examine results of pressure relief calculations:

1. From the Data menu, point to Flowsheeting Options, then Pressure Relief.
2. On the Pressure Relief Object Manager, select the Pressure Relief ID of interest, and click Edit.
3. In the left pane of the Data Browser, click the appropriate results form for the selected Pressure Relief block.

   For steady-state scenarios click the SteadyStateResults form.

   For dynamic scenarios, click the DynamicResults form.

Steady-State Results

Use the SteadyStateResults form to view calculated results for steady state simulations.

The SteadyStateResults form is comprised of two sheets:

Use this sheet	To view
Summary	Input summary, whether code requirements are met, and inlet and tail pipe pressure changes
Property Profiles	Property profiles for points along the relief system

Summary

Use this sheet to view:

- Input summary (Scenario, Relief Device type, and Capacity option)
- Whether code requirements are met
- Actual flow rate through the emergency relief system as calculated by Aspen Plus
- Estimated flow rate you provided on the Setup Scenario sheet.
- Actual and allowed pressure loss through the vessel neck and inlet pipes
- Actual and allowed backpressure in the tail pipes

If code requirements are not met, you should review the status messages. Your system may not meet code requirements because:

- Inlet pipe pressure loss is too high
- Tail pipe pressure loss is too high
- The 97% rule has been violated
- Choke point(s) do not occur at the relief device

For inlet piping, the actual (calculated) loss is the pressure loss through the vessel neck and the inlet pipes computed at 10% overpressure. If this field is blank, it means that the source pressure was greater than 10% overpressure. The allowed pressure loss is the value calculated from the allowed inlet pipe pressure loss as a percentage of differential set pressure as specified on the Setup Rules sheet.

For outlet piping, the actual (calculated) backpressure is computed at 10% overpressure. If this field is blank, it means that the pressure was greater than 10% overpressure or the 97% rule was used instead. The allowed pressure loss is the value computed from the maximum allowed percentage specified on the Setup Rules sheet.

Property Profiles

Use this sheet to view property profiles for points along the emergency relief system. The following properties are shown:

- Status (e.g., OK, or Choked)
- Pressure
- Temperature
- Vapor Fraction

Dynamic Results

Use the DynamicResults form to view calculated results for dynamic simulations. The DynamicResults form is comprised of seven sheets:

Use this sheet	To view
Summary	Input summary, whether code requirements are met, and inlet and tail pipe pressure changes
Parameters	Summary of dynamic results and vessel pressures and temperatures
Vessel	Vessel properties versus operation time
Vent	Property profiles for points along the relief system versus operation time
Accumulator	Accumulator properties versus operation time
X-Y-Z	Vessel, vent, and accumulator mole fractions versus operation time
Vessel Mass	Component mass amounts in vessel per substream versus operation time

Each results sheet is discussed below.

Summary

Use this sheet to view:

- Input summary (Scenario, Relief Device type, and Capacity option)
- Whether code requirements are met
- Initial, final, maximum and allowed vessel pressures
- Initial, final, maximum and allowed vessel temperatures
- Actual and allowed pressure loss through the vessel neck and inlet pipes
- Actual and allowed backpressure in the tail pipes

If requirements are not met, you should review the status messages. If your simulation fails to meet code requirements, possible causes are:

- Inlet pipe pressure loss at 10% over-pressure is too high
- Tail pipe pressure loss at 10% over-pressure is too high
- The 97% rule has been violated
- Choke point(s) do not occur at the relief device
- Vessel pressure goes above the maximum allowed

For inlet piping, the actual (calculated) loss is the pressure loss through the vessel neck and the inlet pipes computed at 10% overpressure. If this field is blank, it means that the source pressure was greater than 10% overpressure. The allowed pressure loss is the value calculated from the allowed inlet pipe pressure loss as a percentage of differential set pressure as specified on the Setup Rules sheet.

For outlet piping, the actual (calculated) backpressure is computed at 10% overpressure. If this field is blank, it means that the pressure was greater than 10% overpressure or the 97% rule was used instead. The allowed pressure loss is the value computed from the maximum allowed percentage specified on the Setup Rules sheet.

Parameters

Use this sheet to view:

- Operation time (the time for which the simulation ran)
- Calculated vessel volume
- Calculated vessel wetted area
- Fire heat input, based upon the wetted area and fire credit factors
- Fire credit factor
- Vent maximum flow (the maximum flow rate through the emergency relief system calculated during the simulation)

Allowed vessel conditions are based upon your input for vessel maximum pressure on the Setup Rules sheet.

Vessel

This results sheet displays a table of vessel properties versus operation time. The following vessel properties are shown:

- Status
- Vent flow
- Pressure
- Temperature
- Vapor fraction
- Total mass

The status column indicates where the choke is, if not at the device. Listed below is an explanation of the possible status symbols:

Status	Meaning
CL	Vent closed
OK	Acceptable choke location and pressure
N	Choke at vessel neck
VF	Choke at valve flange
I1	Choke at Inlet Pipe 1
I2	Choke at Inlet Pipe 2
T1	Choke at Tail Pipe 1

Continued

Status	Meaning
T2	Choke at Tail Pipe 2
SH	Large static head
XT	Excess tail pressure

Vent

This sheet lets you view profiles for points along the relief system versus operation time for the following properties:

- Temperature
- Pressure
- Vapor fraction
- Mass density

Accumulator

The Accumulator sheet lets you view accumulator properties versus operation time. The following accumulator properties are shown:

- Pressure
- Temperature
- Vapor fraction
- Total mass in the accumulator

X-Y-Z

The X-Y-Z sheet lets you view component mole fractions versus operation time for the following phases and locations:

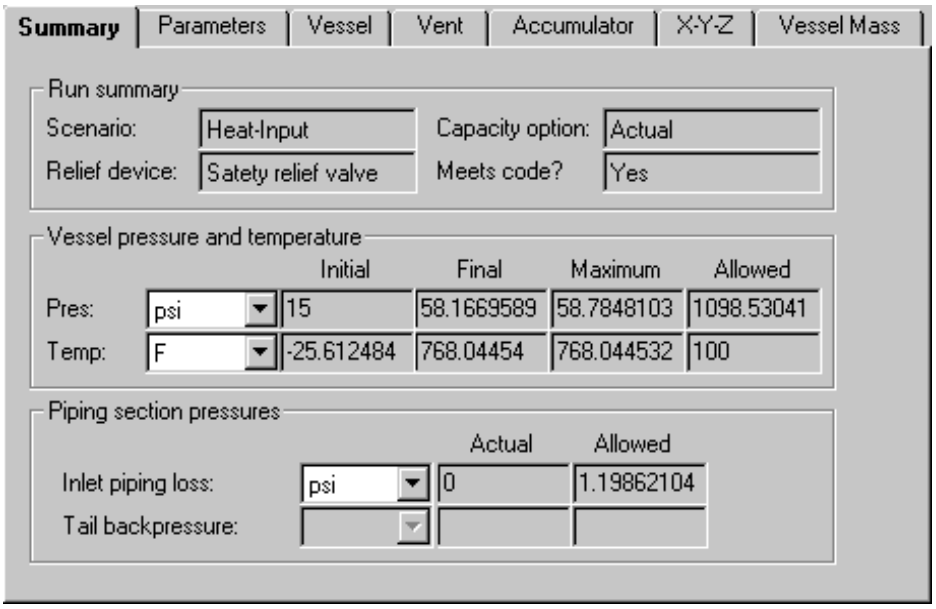
- Overall contents of the vessel (Vessel Total)
- Vapor phase of vessel (Vessel Vapor)
- Overall contents of the relief system (Vent Total)
- Overall contents of the accumulator (Accumulator Total)
- Liquid phase of vessel (Vessel Liquid)

Vessel Mass

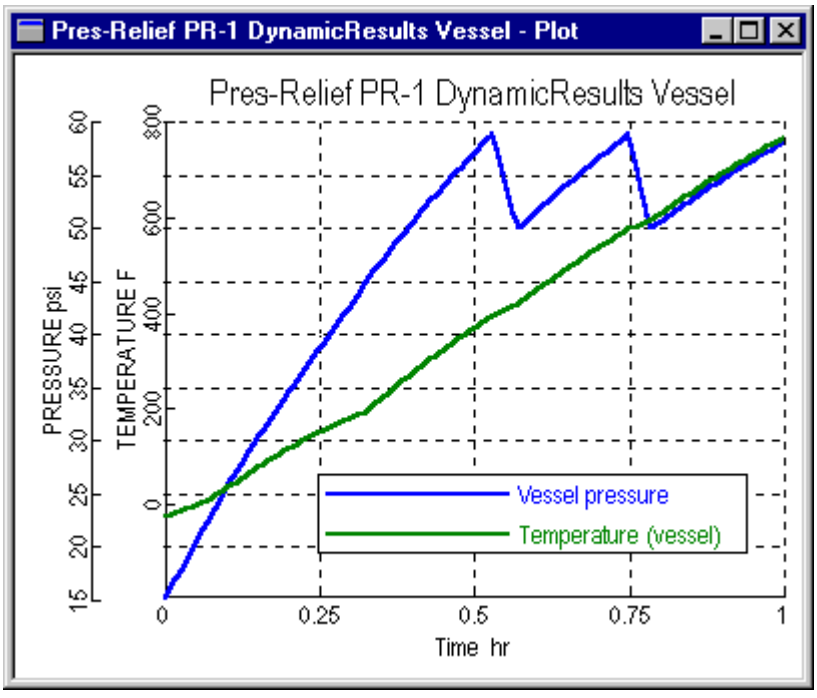
The Vessel sheet lets you view the component mass amounts in the vessel versus operation time for the substream you select.

Example of Dynamic Run of a Pressure Relief System

This example shows the results after a dynamic run of a pressure relief system. The first screen shows the DynamicResults with the Summary sheet displayed:



This plot shows vessel pressure and temperature over time:



❖ ❖ ❖ ❖

34 Inserts

An insert is a partial backup file that you can import into a run at any time. Aspen Plus provides special data packages as inserts which can be used as starting points for building new simulations, or they can be imported into existing simulations. You can create your own inserts for later use.

This chapter discusses this use of files including:

- What is an insert?
- Creating an insert
- Importing inserts
- Creating a property package
- Resolving ID conflicts
- Using electrolyte inserts from the Aspen Plus insert library
- Hiding objects

Creating an Insert

You can use an insert to create a:

- Property package, consisting of component and property definitions
- Standard process unit, such as a crude column and its preheat train

You can create your own inserts, or you can import inserts from the Aspen Plus library of inserts. For more information, see Using Electrolyte Inserts From the Aspen Plus Insert Library on page 34-6.

To create an insert, you need to save a backup file containing the information you want in your insert:

1. Begin with a run that has all of the input for the insert defined. (The run does not have to be complete.) This can consist of any possible simulation input such as components, properties, streams, blocks, flowsheeting options, model analysis tools, etc.
2. From the File menu, click Export.
3. In the Save As Type box, select Aspen Plus Backup Files (*.bkp).
4. Enter a path and a filename for the backup file that you want to contain the insert.
5. Click Save.

   You can import the backup file you created into any run.

Importing Inserts

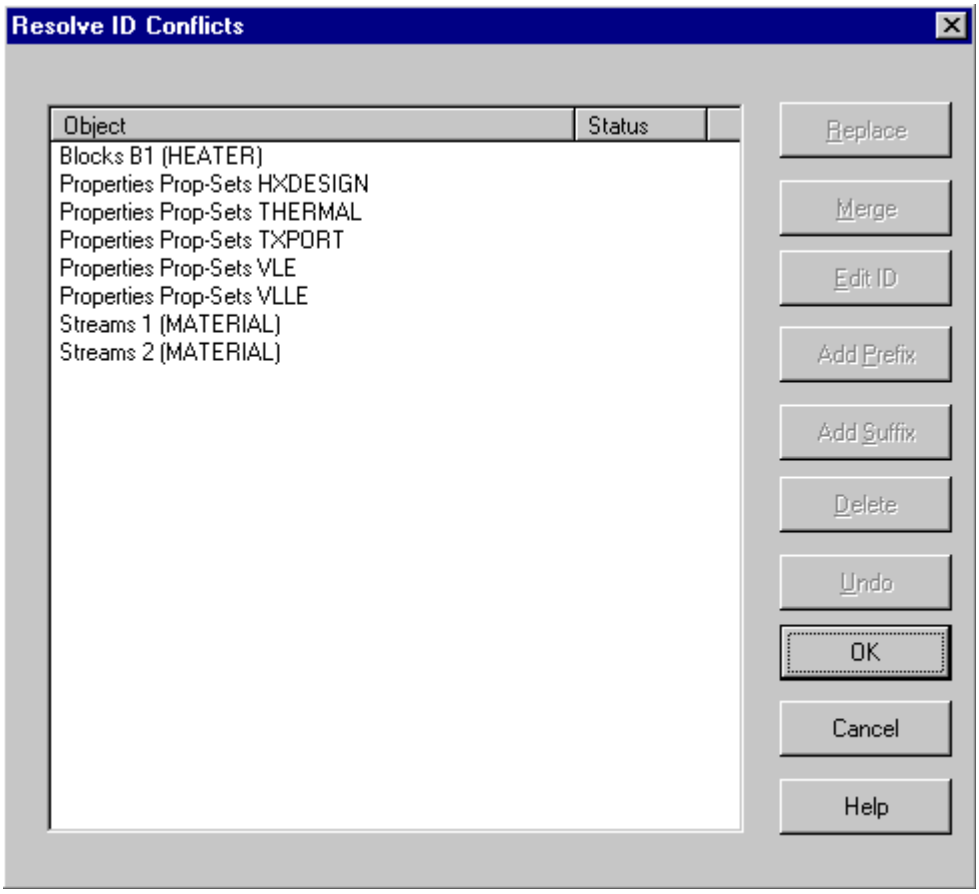
To import an insert into an existing Aspen Plus simulation:

1. With the existing simulation active in the Aspen Plus main window, from the File menu, click Import.
2. In the Save As Type box, select Aspen Plus Backup Files (*.bkp).
3. On the Import dialog box, locate the insert, select it, and click Open.
4. If the Resolve ID Conflicts dialog box appears, see Resolving ID Conflicts on page 34-3.

   After importing the insert into your existing run, your simulation will contain the input from both files.

Resolving ID Conflicts

When you import one file into another, some imported objects may have the same IDs as objects in the existing run. When this happens, Aspen Plus displays the Resolve ID Conflicts dialog box, which lists all objects that have matching IDs in the two files.



Use the Resolve ID Conflicts dialog box to resolve ID conflicts by one of the following methods:

Method	Procedure
Replace existing objects	1. Select one or more objects. 2. Click Replace. Aspen Plus deletes the objects in the current run and replaces them with the objects being imported.
Merge new objects with existing objects	1. Select one or more objects. 2. Click Merge. Aspen Plus merges specifications for inserted objects with those of objects in the current run. If both objects have values for the same specification, the inserted object overrides the object in the current run.
Edit IDs directly	1. Select one object at a time. 2. Click Edit ID. 3. In the Object Name dialog box, specify a new ID for the object.
Add a prefix or suffix to the existing IDs	1. Select one or more objects. 2. Click Add Prefix or Add Suffix. 3. In the Prefix (or Suffix) dialog box, enter characters to be added to the IDs in the existing run.
Delete imported objects	1. Select one or more objects. 2. Click Delete. Aspen Plus deletes the selected objects from the imported run, leaving those objects in the existing run unchanged.

When finished resolving each ID conflict, click OK.

Merged objects must be of the same type. For example, you can merge two RadFrac blocks, but not a RadFrac block with a Flash2 block.

Example of Importing an Insert and Resolving ID Conflicts

Simulate a distillation column with two different feeds in the same run. The distillation column specifications are identical.

1. Create the flowsheet for the first feed and complete all specifications for the problem.
2. From the File menu, click Save As.
3. In the Save As Type box, specify Aspen Plus Backup Files (*.bkp). Specify a name for the file in the File Name box. Click OK.
4. From the File menu, click Import.
5. In the Files of Type box, select Aspen Plus Backup Files (*.bkp). Locate and select the file you just saved. Click OK.
6. In the Resolve ID conflicts dialog box, select the block and streams. Hold down the Ctrl key while clicking on each item. Then click Add Suffix.

7. In the Suffix dialog box, enter -2, and click OK.
8. The Resolve ID Conflicts dialog box now shows the new IDs for the inserted objects.
9. Select all the remaining objects in the Resolve ID Conflicts dialog box and click Delete.
10. In the Resolve ID Conflicts dialog box, click OK.

    Aspen Plus adds the new block, and streams to the flowsheet.

Example of Copying a Block from One Run to Another

This example assumes that two runs have identical Sep2 component splitters. The block ID and the inlet and outlet stream IDs for Sep2 are the same in both runs. Replace Sep2 with an identical RadFrac rigorous distillation model in both runs.

1. Replace Sep2 with RadFrac in the first run, and complete the specifications.
2. From the File menu, click Save As and save the file as an Aspen Plus Backup File (*.bkp).
3. Open the second run.
4. From the File menu, click Import. Select the file you saved in step 2 and click OK.
5. In the Resolve ID Conflicts dialog box, select the RadFrac block and all the streams listed. Click on the Replace button.
6. Select all the remaining objects in the Resolve ID Conflicts dialog box and click Delete.
7. In the Resolve ID Conflicts dialog box, click OK.

   The new RadFrac block, with all of its specifications, now replaces the Sep2 block in the flowsheet.

Creating a Property Package

To create a property package:

1. Begin with a run that has all of the input for the property package defined, including all components and properties specifications. (The run does not have to be complete.)

Typically this would include:

- Components specifications
- Henry-Comps specifications, if defined
- Chemistry specifications, if defined
- Properties specifications
- Property method definitions other than built-in property methods
- Any Properties Parameters objects with data specified
- Any units sets, other than SI, MET, or ENG, used by any of the forms in the property package
- Any property sets you want to include

2. From the File menu, click Export.
3. In the Export dialog box, enter a path and a filename for the backup file that you want to contain the property package.
4. Click Save.

   You can import the backup file you created into any run.

For example, suppose you develop a property package for ethanol-water using the NRTL property method. Specify the following information in Aspen Plus, and save the specifications as a backup file:

- Components Specifications Selection sheet
- Properties Specifications Global sheet
- Properties Parameters Binary Interaction NRTL-1 form

Using Electrolyte Inserts From the Aspen Plus Insert Library

To use an insert from the Aspen Plus library:

1. From the File menu, click Import.
2. On the Import dialog box, click the Favorites button on the toolbar.
3. In the Favorites folder, double-click the Elecins folder.
4. Select an insert from the list and click Open.
5. If the Resolve ID Conflicts dialog box appears, see Resolving ID Conflicts on page 34-3.

   Tip To see a description on the insert, use the Preview button on the Import dialog box toolbar.

Tip To view in detail the contents of an insert before using it, follow the procedure above, except open the insert using File Open, instead of File Import. Then use the Data Browser to see what input is defined in the insert and to look at the insert contents.

Hiding Objects

You can use the Hide feature to temporarily remove optional objects from a simulation, without deleting them. For example, you can hide a design specification when you don't want it to be applied to the simulation.

You cannot hide:

- Global specifications, such as the Setup Specifications and Properties Specifications forms
- Components
- Blocks and streams
- Properties Parameters and Molecular Structure objects

To hide objects:

1. Display the Object Manager for the type of object you want to hide.
2. Select one or more objects you want to hide. If the Hide button is dim, you cannot hide this type of object.
3. Click the Hide button.

   Aspen Plus removes the selected objects from the Object Manager list. They are no longer part of the problem definition.

Revealing Objects

To reveal (unhide) objects:

1. Open the Object Manager for the type of object you want to reveal.
2. Click the Reveal button.

   Note The Reveal button will only be active if there are hidden objects.

3. On the Reveal dialog box, select the hidden objects you want to reveal, and click OK.

   If there are no ID conflicts, Aspen Plus restores the objects to the problem definition, and they are displayed on the Object Manager.

If the specifications for a hidden object are inconsistent with the current problem definition (for example, if a referenced stream no longer exists), the object will be incomplete. Use Next to find out what you must do to complete the input.

If there are ID conflicts (if a hidden object has the same ID as an object in the current problem definition), the Resolve ID Conflicts dialog box appears.

Tip Use the Remove button on the Reveal dialog box to permanently delete hidden objects from the simulation.

35 Creating Stream Libraries

This chapter describes how to create stream libraries. For information on how to retrieve information from a stream library for use in a simulation, see Chapter 9.

- Creating or modifying a stream library
- STRLIB command summary
- Description of STRLIB commands

You can retrieve information about stream composition and conditions from a stream library, instead of entering this data on stream forms.

This table shows what you can do with the stream library capability:

To	Do This
Create a library of frequently used feed streams	Store the composition and conditions of frequently used feed streams in a stream library. Retrieve this information from different models in a simulation without re-entering it.
Transfer stream information from one simulation to another	Simulate one section of your flowsheet, store the outlet streams in a library, and retrieve the information in another simulation. Or use a stream library to share information between two groups that are simulating different sections of a process.
Initialize tear streams	Store final tear stream values from a simulation in a library. When you simulate another case, retrieve the desired values as an initial guess for the tear stream. If you do not know which stream will be chosen as the tear stream, store all the streams from the first simulation and retrieve all the streams in the new run.
Isolate a block from a large flowsheet	Store streams for a large flowsheet in a library. Retrieve and analyze one block from the stored flowsheet and simulate the block by itself, perhaps with higher diagnostics or at different conditions. This eliminates re-entering the stream information for the isolated block run.

Creating or Modifying a Stream Library

You can:

- Create your own stream library
- Use a stream library created and maintained by your Aspen Plus system administrator

To create or modify a stream library, use STRLIB, a program delivered with Aspen Plus. Every Aspen Plus run produces a summary file, which contains all the results of the simulation. STRLIB copies stream results from an Aspen Plus summary file into a stream library. You can store data from any number of Aspen Plus runs in one stream library.

A stream library is organized in cases. You identify a stream by the:

- Stream name
- Case to which it belongs

Each case usually corresponds to one Aspen Plus run. However, you can store streams from more than one Aspen Plus run in a single case. You can also store data from a single run in several cases.

Using STRLIB, you enter commands to create or modify a stream library. The most common use of STRLIB involves these steps:

1. Opening a summary file using the OPEN command.
2. Establishing the case where you want to copy streams, using the OPEN or CASE commands.
3. Adding or replacing streams in the library, using the ADD or REPLACE commands.

Running STRLIB Interactively

To run STRLIB interactively, enter this command at the operating system prompt:

strlib *libname*

Where *libname* is the stream library you want to create or modify. The stream library name can be up to eight characters long.

Library files have the extension *.slb.

When the STRLIB> prompt appears, you can enter commands.

STRLIB prompts you for each command.

Running STRLIB in Batch Mode

You can run STRLIB non-interactively to create or update a stream library. Running STRLIB in batch mode automatically adds all the streams from the summary file produced by an Aspen Plus run.

To run STRLIB in batch mode, enter this command at the operating system prompt:

STRLIB ***libname*** ***runid*** **[case]**

libname is the name of the library you want to create or modify. If the library does not exist, it will be created and initialized to contain 10 cases.

runid is the name of the Aspen Plus summary file from which you want to transfer streams.

case is the case name where you want to add streams. If you do not specify the case name, STRLIB uses the runid from the summary file as the case name.

The batch mode STRLIB command is equivalent to the following sequence of commands when you run STRLIB interactively:

```
STRLIB>  OPEN    runid    [case]
STRLIB>  ADD     ALL
STRLIB>  EXIT
```

STRLIB Command Summary

This table lists a summary of the commands you can use in STRLIB. The commands are described in more detail in this chapter.

You can abbreviate any STRLIB command. Enter all commands at the STRLIB> prompt, typing enough letters to identify the command as unique. The first one or two letters are usually enough.

STRLIB Commands

Command	Description
ADD	Add a stream to the library.
CASE	Change the current case.
DELCASE	Delete a case from the library.
DELSTREAM	Delete a stream from the library.
DIRECTORY	List the cases in the library and the streams in a case.
DUMP	Write stream information to a file.
END	End STRLIB and update the library.
EXIT	End STRLIB and update the library.
HELP	Display interactive help on STRLIB commands.
INITIALIZE	Initialize a library.
LIST	List streams in the summary file.
LOAD	Load stream information from a dump file.
OPEN	Open a summary file.
PACK	Pack (compress) the library.
RENAME	Rename a stream in the library.
REPLACE	Replace a stream in the library.
QUIT	End STRLIB without updating the library.

Description of STRLIB Commands

ADD

The ADD command copies a stream from an Aspen Plus summary file to the library. The stream to be copied must not already exist in the current case of the library. (Use the REPLACE command to replace streams that already exist.)

Specify ADD ALL to copy all streams from a summary file to the library.

Syntax:

ADD $\left\{\begin{matrix} stream-ID \\ ALL \end{matrix}\right\}$

CASE

The CASE command changes the current case. Streams in the stream library are organized into cases. You establish a case when you open a summary file (using the OPEN command) or when you use the CASE command. For more information about cases, see Creating or Modifying a Stream Library on page 35-2.

The ADD, DELSTREAM, DUMP, RENAME, and REPLACE commands apply to streams in the current case.

Syntax:

CASE ***casename***

DELCASE

The DELCASE command deletes a case from the library. All streams in the case are deleted.

Syntax:

DELCASE ***casename***

DELSTREAM

The DELSTREAM command deletes a stream from the current case.

If you delete many streams from a library, you should use the PACK command to recover the deleted space. See the PACK command description, this section.

Syntax:

DELSTREAM ***stream-id***

DIRECTORY

The DIRECTORY command lists the cases and streams stored in the stream library. If you do not specify a case in the DIRECTORY command, STRLIB lists all the cases in the library and the number of streams in each case. If you specify a case, STRLIB lists only the streams in that case.

Syntax:

DIRECTORY [***casename***]

DUMP

The DUMP command writes the information about a stream stored in the library. STRLIB prompts you to specify whether you want to write to the terminal or to a file. If you write to a file, STRLIB prompts you for the filename. Specify DUMP ALL to dump all streams from all cases in a library.

Use DUMP if you want to:

- View the information for a stream
- Transfer information from one library to another

If you need to reinitialize a library to increase the maximum number of cases that can be stored, use the DUMP ALL command first to save the contents of the library. To restore the information, use the LOAD command.

Syntax:

DUMP $\left\{\begin{matrix} stream-ID \\ ALL \end{matrix}\right\}$

END

The END command ends the STRLIB session. The stream library is updated with all the changes you made during the session. The END and EXIT commands are synonymous. (Use the QUIT command to end STRLIB without updating the library, so changes made during the session are not saved.)

Syntax:

END

EXIT

The EXIT command ends the STRLIB session. The stream library is updated with all changes you made during the session. The END and EXIT commands are synonymous. (Use the QUIT command to end STRLIB without updating the library, so changes made during the session are not saved.)

Syntax:

EXIT

HELP

The HELP command activates the interactive help system, so you can obtain help for STRLIB commands.

Syntax:

HELP [***command***]

INITIALIZE

The INITIALIZE command destroys all data in a stream library. Use it only when creating a library or after using the DUMP ALL command.

The INITIALIZE command initializes a new stream library. You must specify the maximum number of cases the library will contain. Enter the INITIALIZE command before performing any operations on a new stream library.

Syntax:

INITIALIZE ***numcase***

LIST

The LIST command lists the streams in the current summary file.

Syntax:

LIST

LOAD

The LOAD command loads information from a dump file created with the DUMP command. STRLIB loads all cases and streams.

Syntax:

LOAD ***filename***

OPEN

The OPEN command opens a summary file, so that streams from an Aspen Plus run can be transferred to the library. If you do not specify a case name, STRLIB uses the RUNID from the summary file as the case name.

Syntax:

OPEN ***filename*** [***casename***]

PACK

The PACK command packs the stream library to recover blank spaces created when streams are deleted. The PACK command is necessary only if you delete many streams from a library and want to recover unused file space.

Syntax:

PACK

RENAME

The RENAME command renames a stream in the library. RENAME applies only to the current case.

Syntax:

RENAME ***oldname newname***

REPLACE

The REPLACE command replaces a stream in the current case with a stream of the same name from a summary file. If the stream does not exist in the library, STRLIB adds it.

Specify REPLACE ALL to copy all streams from a summary file to the library, overwriting any streams of the same name that exist in the library.

Syntax:

REPLACE $\left\{\begin{matrix} stream-ID \\ ALL \end{matrix}\right\}$

QUIT

The QUIT command ends the STRLIB session. The stream library is not updated with any changes made during the current STRLIB session. (Use the END or EXIT commands to end STRLIB and update the stream library with the changes made during the session.)

Syntax:

QUIT

Example of Creating a Library with Two Cases

Create a stream library that can contain two cases. Add streams S1 and S2 from summary file RUN1. Also add all the streams from summary file RUN2.

```
STRLIB>   INITIALIZE 2
STRLIB>   OPEN RUN1.SUM
STRLIB>   ADD S1
STRLIB>   ADD S2
STRLIB>   OPEN RUN2.SUM
STRLIB>   ADD ALL
STRLIB>   END
```

The library will be organized in two cases, RUN1 and RUN2.

Example of Creating a Library with One Case

Create a stream library that can hold five cases. Create one case, PROJECT. Add streams PROD1 and PROD2 from summary file RUN1, and stream FEED1 from summary file RUN2.

```
STRLIB>   INITIALIZE 5
STRLIB>   OPEN RUN1.SUM
STRLIB>   CASE PROJECT
STRLIB>   ADD PROD1
STRLIB>   ADD PROD2
STRLIB>   OPEN RUN2.SUM
STRLIB>   CASE PROJECT
STRLIB>   ADD FEED1
STRLIB>   END
```

The OPEN command creates a new case if a case name is not specified. The library must be initialized to hold at least three cases. The preceding set of commands will create two empty cases named RUN1 and RUN2.

To avoid creating the empty cases, use the case name in the OPEN command.

```
STRLIB>   INITIALIZE 5

STRLIB>   OPEN RUN1.SUM PROJECT

STRLIB>   ADD PROD1

STRLIB>   ADD PROD2

STRLIB>   OPEN RUN2.SUM PROJECT

STRLIB>   ADD FEED1

STRLIB>   END
```

❖ ❖ ❖ ❖

36 Stream Summary Formats

Aspen Plus allows you to customize stream reports and tables using Table Format Files (TFFs). Using TFF language, you can customize the:

- Results Summary Streams form and block StreamResult sheets to your own format, for analyzing your simulation results
- Stream table in a process flow diagram (PFD), to meet your company's standards

A Table Format File contains easy-to-understand language that you can use to:

- Display a selected list of stream properties of interest, in a specified order
- Add or change labels for stream properties
- Manipulate the format of stream property values (for example, scaling, normalization, units conversion, and trace cut-off)

This chapter explains how to create and use TFFs. Sample TFFs are provided. Read this chapter if you plan to customize your stream summary form or the stream table in your process flow diagram (PFD).

Topics include:

- About the Aspen Plus TFFs
- Creating a TFF
- Basic Stream Result Properties

About the Aspen Plus TFFs

Aspen Plus provides several Table Format Files in your system directory. If you installed Aspen Plus in the default directory, your system directory is Program Files\AP10UI\xeq.

By default Aspen Plus displays the stream summary and stream table based on the built-in TFF for the Application Type you chose when creating your simulation.

Choose a stream format to display your stream report from:

- The Stream Format fields on the Setup Specifications Stream Report
- The Format field on the ResultsSummary Streams Material sheet or the block StreamResults Material sheet

All of the TFF files located in either the system directory or in the working directory are displayed in the list. You can modify any Aspen Plus TFF or create your own. TFF files should be placed either in your working directory or in the system directory.

Aspen Plus uses the TFF you select in either field for all Results Summary Streams sheets you display, until you select another TFF.

To select a stream format:

1. Move to the Stream Format field of the Setup Specifications Stream Report sheet, the ResultsSummary Streams Material sheet, or the block StreamResults Material sheet.
2. Click the list and scroll through the options, looking at the descriptions of each TFF.
3. Select a TFF. If you are using built-in TFFs, it is recommended that you select one of the TFFs for your Application Type. For example, if you are using one of the Petroleum Application Types, choose a TFF beginning with Petro.

It is not necessary to re-run the simulation in order to see the results in another format.

Creating a TFF

You can:

- Edit the TFFs provided with Aspen Plus to customize your stream summary and stream tables. These files are located in the Program Files\AP10UI\xeq directory if you installed Aspen Plus in the default directory.
- Use TFF language to create your own TFF.

The sections that follow describe how to edit or create a new TFF. You may also want to refer to the example TFFs included at the end of the chapter.

TFF File Format and Options

Use TFF language to customize your stream summary or stream table. Follow these rules:

- TFF sentences are not case sensitive.
- Any line beginning with a semi-colon in column 1 is treated as a comment line.
- The ampersand (&) is used to continue a line.

The format for the TFF and a description of the format follow.

TITLE = value
STREAMS=value
STREAM-ID-LABEL = value
SOURCE-LABEL = value
DEST-LABEL = value
PHASE-LABEL = value
BEGLOOP SUBSTREAM = value
ENDLOOP
DISPLAY *qualifier optional qualifier=value option=value*

Qualifiers:

ALL ONLY REMAIN

Optional qualifiers:

SUBSTREAM COMPS PHASE BASIS TEMP PRES LVPCT COMP-ATTR SUBS-ATTR

Options:

FORMAT PPM PPB TRACE TRACE-LABEL ZERO-LABEL MISSING-LABEL PROP-HEADER COMPS-HEADER SUBSTREAM-HEADER PB-HEADER TEMP-HEADER PRES-HEADER LVPCT-HEADER COMP-ATTR-HEADER SUBS-ATTR-HEADER COMP-ATTR-ELEM SUBS-ATTR-ELEM

PROP **prop-name** *qualifier=value* *option=value*

Optional qualifiers:

SUBSTREAM **COMPS** **PHASE** **BASIS** **TEMP** **PRES** **LVPCT** **COMP-ATTR** **SUBS-ATTR**

Options:

FORMAT **PROP-LABEL** **UNITS** **UNITS-LABEL** **NORMALIZE** **SCALE** **SCALE-LABEL** **PPM** **PPB** **TRACE** **TRACE-LABEL** **ZERO-LABEL** **MISSING-LABEL** **MW** **BP** **MW-BP-FORMAT** **HEADER** **PROP-HEADER** **COMPS-HEADER** **SUBSTREAM-HEADER** **PB-HEADER** **TEMP-HEADER** **PRES-HEADER** **LVPCT-HEADER** **COMP-ATTR-HEADER** **SUBS-ATTR-HEADER** **COMP-ATTR-ELEM** **SUBS-ATTR-ELEM**

TEXT "Text enclosed in double quotes"

TFF Sentences

This section describes each sentence that you can use in a TFF.

TITLE

Title for the stream table. The TITLE line must be the first non-comment line in the TFF. If you choose to wrap the stream table, TITLE is not repeated. TITLE is not displayed in the Results Summary Streams form.

TITLE=YES ..... The title you specified on the Setup Specifications form is used. If no title is specified on Setup Specifications, the title Heat and Material Balance Table is used.

TITLE=NO ..... No title is displayed. (Default)

TITLE="string" ..... The string (up to 64 characters) enclosed in double quotes is used as the title for the stream table.

STREAMS

Use to define a set of streams and their order on the Results Summary Streams form and in the stream table.

STREAMS=sid-list ..... List of stream IDs

If the STREAMS statement is not in your TFF, all streams are displayed in alphanumeric order. (Default)

You can interactively select the streams and the order they are displayed on the Results Summary Streams form.

STREAM-ID-LABEL

Label for the stream ID row in the stream table. See Header Sentence Order in the Stream Table, this chapter.

STREAM-ID-LABEL=YES The label Stream ID is used.

STREAM-ID-LABEL=NO .. The Stream ID row is not displayed. (Default)

STREAM-ID-LABEL="string" The string (up to 20 characters) in double quotes is used as the label.

SOURCE-LABEL Label for the source block row in the stream table. See Header Sentence Order in the Stream Table, this chapter.

SOURCE-LABEL=YES .... The label From is used.

SOURCE-LABEL=NO ..... The source block row is not displayed. (Default)

SOURCE-LABEL="string" The string (up to 20 characters) enclosed in double quotes is used as the label.

DEST-LABEL Label for the destination block row in the stream table. See Header Sentence Order in the Stream Table, this chapter.

DEST-LABEL=YES ..... The label To is used.

DEST-LABEL=NO ..... The destination block row is not displayed. (Default)

DEST-LABEL="string" .... The string (up to 20 characters) enclosed in double quotes is used as the label.

PHASE-LABEL Label for the phase row for the MIXED substream. See Header Sentence Order in the Stream Table, this chapter.

PHASE-LABEL=YES ..... The label Phase is used.

PHASE-LABEL=NO ..... The phase row is not displayed. (Default)

PHASE-LABEL="string" .. The string (up to 20 characters) enclosed in double quotes is used as the label.

BEGLOOP, ENDLOOP Optional sentences that enable you to control the display of properties when there are two or more substreams. You can define groups of DISPLAY sentences by enclosing them between pairs of BEGLOOP and ENDLOOP sentences. Aspen Plus displays properties specified by the enclosed DISPLAY sentence for one substream at a time, looping through all requested substreams.

SUBSTREAM ..... List of substreams to loop through

SUBSTREAM=ssid-list..... List of substream IDs

SUBSTREAM=ALL ..... All substreams

DISPLAY Used to control the display of the stream properties. DISPLAY is usually used in conjunction with one or more PROP sentences. PROP sentences can control the order and display of individual properties. You can have any number of DISPLAY sentences in a TFF.

DISPLAY ALL ..... Display all stream properties identified by the specified qualifiers. (See Qualifier Descriptions for DISPLAY and PROP, this chapter.) Use the format described by the options. (See Option Descriptions for DISPLAY and PROP, this chapter.)

DISPLAY ONLY ..... Display only the stream properties specified by the PROP sentences following the DISPLAY ONLY sentence. Use the order of the PROP sentences, with the specified qualifiers and options.

DISPLAY REMAIN ..... Display the remaining stream properties (those not already specified by DISPLAY or PROP sentences). Identify stream properties by the specified qualifiers, in the format described by the options.

PROP

Used to control the display of an individual property. When used in conjunction with DISPLAY ONLY, PROP sentences specify the order in which the properties are displayed. See Qualifier Descriptions for DISPLAY and PROP, and Option Descriptions for DISPLAY and PROP, this chapter.

prop-name.....Stream property. This is the Prop-Set name for a stream property (for example, MOLEFLMX for total mole flow). Prop-Set names are listed in the Property field on the Prop-Sets form. A property displayed in the stream summary or a stream table must be one of the basic stream result properties listed in the table on page 36-8 , or the property must be included in a prop-set ID in the Property sets field on the Setup Specification Stream Report form. All such properties appear in standard stream reports for your run.

If your TFF requests a stream property that is not in listed in the table or included in a prop-set ID, Aspen Plus does not display the property.

The options specified with the PROP sentence are combined with the DISPLAY options when Aspen Plus displays the properties. If the same option is specified for PROP and the preceding DISPLAY sentence, the PROP specification is used.

TEXT

Allows you to insert a text line within the side label of your stream table. Enclose text in double quotes. To insert a blank line, use a pair of double quotes with a space between them.

The following table lists the basic stream result properties.

Basic Stream Result Properties

Prop-Name	Description
MOLEFLOW†	Component mole flow
MASSFLOW†	Component mass flow
VLSTD†	Component standard volume flow
MOLEFRAC†	Component mole fraction
MASSFRAC†	Component mass fraction
VLSTDFR†	Component standard volume fraction
MOLEFLMX	Total mole flow
MASSFLMX	Total mass flow
VOLFLMX	Total volume flow
TEMP	Temperature
PRES	Pressure
VFRAC	Vapor fraction
LFRAC	Liquid fraction
SFRAC	Solid fraction
HMX	Enthalpy (in mole, mass, and flow basis)
SMX	Entropy (in mole and mass basis)
RHOMX	Density (in mole and mass basis)
MWMX	Average molecular weight
COMP-ATTR	Component attributes
SUBS-ATTR	Substream attributes
CMASS_TIME††	Component mass flow rate during actual operation
CMOLE_TIME††	Component mole flow rate during actual operation
CVOL_TIME††	Component standard volume flow rate during actual operation
CMASS_CYCLE††	Component mass per cycle
CMOLE_CYCLE††	Component mole per cycle
CVOL_CYCLE††	Component standard volume per cycle
CYCLE_TIME††	Cycle time
OPER_TIME††	Operation time

Continued

Basic Stream Result Properties (continued)

Prop-Name	Description
NTRAIN††	Number of trains
MASS_TIME††	Total mass flow rate during actual operation
MOLE_TIME††	Total mole flow rate during actual operation
VOL_TIME††	Total volume flow rate during actual operation
ENTH_TIME††	Total enthalpy flow rate during actual operation
MASS_CYCLE††	Total mass per cycle
MOLE_CYCLE††	Total mole per cycle
VOL_CYCLE††	Total volume per cycle
ENTH_CYCLE††	Total enthalpy per cycle

† *Property is available if the corresponding option is selected on the Setup.Main form.*

†† *Batch stream properties. Displayed only if you use the Report.Batch-Operation form.*

Qualifier Descriptions for DISPLAY and PROP

This section describes the qualifiers you can use in both DISPLAY and PROP sentences. You can think of the combined DISPLAY and PROP qualifiers as a property filter. Any property in the stream report that passes the filter is displayed.

The qualifiers listed are all possible specifications for a Prop-Set property, except UNITS. See Option Descriptions for DISPLAY or PROP, this chapter. The only qualifiers that apply to the basic stream report properties are SUBSTREAM and COMPS. The basic stream result properties do not display if the PHASE or BASIS qualifiers are set to any value other than the default (ALL).

SUBSTREAM.....Substreams for which the property is to be displayed

SUBSTREAM=ssid-list.... List of substream IDs

SUBSTREAM=ALL ..... All substreams (Default)

COMPS .....Components for which the property is to be displayed

COMPS=cid-list ..... List of component IDs

COMPS=ALL..... All components (Default)

PHASE .....Phase for which the property is to be displayed

PHASE=V ..... Vapor

PHASE=L ..... Total liquid

PHASE=L1 ..... 1st liquid

PHASE=L2 ..... 2nd liquid

PHASE=S ..... Solid

PHASE=T ..... Total mixture

PHASE=ALL ..... All phases of the property in the stream report (Default)

BASIS.....Basis for which the property is to be displayed

BASIS=WET..... Includes water

BASIS=DRY ..... Excludes water

BASIS=ALL..... All bases of the property in the stream report (Default)

TEMP.....Temperatures for which the property is to be displayed

TEMP=list..... List of temperatures

TEMP=ALL..... All temperature values of the property in the stream report (Default)

PRES.....Pressures for which the property is to be displayed

PRES=list..... List of pressures

PRES=ALL ..... All pressure values of the property in the stream report (Default)

LVPCT.....Liquid volume percents for which the property is to be displayed

LVPCT=list..... List of liquid volume percents

LVPCT=ALL ..... All liquid volume percent property values in the stream report (Default)

COMP-ATTR .....Component attributes to be displayed

COMP-ATTR=cattr-list .... List of component attributes

COMP-ATTR=ALL ..... All component attributes (Default)

SUBS-ATTR .....Substream attributes to be displayed

SUBS-ATTR=sattr-list..... List of substream attributes

SUBS-ATTR=ALL ..... All substream attributes (Default)

Option Descriptions for DISPLAY and PROP

This section describes the options for the DISPLAY and PROP sentences. These options control the display, side label, and units of a property value.

FORMAT ..... Stream property value display format string, enclosed in double quotes. (Default=customized G format designed to show maximum precision) See Formats for Numbers, this chapter. Applies to both DISPLAY and PROP sentences.

PROP-LABEL ..... Stream property label to override the Aspen Plus property label. Can be used, for example, to replace the Aspen Plus property name MUMX with the label Viscosity. Applies to PROP sentence only. May be truncated in the stream summary, but displays in full in the stream tables.

UNITS ..... Stream property value units of measurement (Setup.Units-Set1, Setup.Units-Set2 and Setup.Units-Set3 forms), enclosed in double quotes. Property value is converted to your specifications. A stream property may be available in more than one type of unit. For example, enthalpy for a stream may have units-types of mole-enthalpy, mass-enthalpy, and enthalpy-flow. In this case the specified units define both the units and the units types to be displayed. If no units are specified, the stream property is displayed in all available units types. The units specification is overridden if you select a units-set on the Results Summary Streams form, but the units-types selection is sustained. Applies to PROP only. (Default=global out-units for basic stream result properties and local units for each additional prop-set).

UNITS-LABEL ..... Label for units of measurement. The Units label is a character string enclosed in double quotes. Overrides the Aspen Plus Units label. Can be used, for example, to print the units label in lowercase characters. Applies to PROP sentence only, and only if the UNITS qualifier is used.

NORMALIZE ..... Normalization flag for component flow or fraction properties

NORMALIZE=YES ..... Normalize property values. See The NORMALIZE Option, this chapter.

NORMALIZE=NO ..... Do not normalize values. (Default)

Applies to DISPLAY and PROP sentences

SCALE ..... Scale factor. Property value is divided by scale factor before it is displayed. Used to reduce the magnitude of printed values. You must also specify SCALE-LABEL. Applies to PROP sentence only.

SCALE-LABEL ..... Scale factor label enclosed in double quotes. Appears in front of the units label or UNITS-LABEL you supply. Applies to PROP sentence only.

PPM ..... Parts per million cut-off value. Property values below the specified number are displayed as PPM. For example, if you specify PPM=1E-3, any property values smaller than 0.001 are displayed as 1 PPM to 999 PPM. Applies only to component flow or fraction properties. See The NORMALIZE Option and PPM, PPB and TRACE Options, this chapter. Applies to both DISPLAY and PROP sentences.

PPB ..... Parts per billion cut-off value. Property values below the specified number are displayed as PPB. For example, by specifying PPB=1E-6, property values smaller than 0.000001 are displayed as 1 PPB to 999 PPB. Applies only to component flow or fraction properties. See The NORMALIZE Option and PPM, PPB, and TRACE Options, this chapter.

Applies to both DISPLAY and PROP sentences.

TRACE .....Trace cut-off value. Property values smaller than the specified cut-off value are not displayed. The character string specified by TRACE-LABEL is displayed instead. Applies to both DISPLAY and PROP sentences. See PPM, PPB, and TRACE Options, this chapter.

TRACE-LABEL .....Trace symbol for displaying trace value, enclosed in double quotes (Default=blank). Applies to both DISPLAY and PROP sentences.

ZERO-LABEL.....Label for zero value enclosed in double quotes (Default=0.0). Applies to both DISPLAY and PROP sentences.

MISSING-LABEL.....Label for property values not calculated, enclosed in double quotes (Default=blank). Applies to both DISPLAY and PROP sentences.

MW .....Molecular weight display. Applies to component-dependent properties only. Displays in the units column in the stream summary or stream table.

MW=YES .....Display molecular weight next to the component ID.

MW=NO .....Do not display molecular weight. (Default)

Applies to PROP sentence only.

BP.....Boiling point display. Applies to component-dependent properties only. Displays in the units column in the stream summary or stream table.

BP=YES.....Display boiling point next to the component ID.

BP=NO .....Do not display boiling point. (Default)

Applies to PROP sentence only.

MW-BP-FORMAT ..... Molecular weight or boiling point format string, enclosed in double quotes (Default=%.0f). See Formats for Numbers, this chapter. Used when MW=YES or BP=YES.

Applies to PROP sentence only.

HEADER ..... Header above a property

HEADER="string" ..... The specified string of up to 20 characters is displayed in the line above the property side label. (Default: no header)

Applies to PROP sentence only.

PROP-HEADER ..... Property header

PROP-HEADER=YES ..... Property label is displayed. (Default)

PROP-HEADER=NO ..... Property label is not displayed.

PROP-HEADER="string". The specified string of up to 20 characters is used as the property label and overrides the PROP-LABEL specification. You can use the TFF variables @PROP (property label) and @UNITS (units label) within this string. Can be truncated in the stream summary but displays in full in the stream table.

Applies to both DISPLAY and PROP sentences.

COMPS-HEADER ..... Component header. Used with component-dependent property only.

COMPS-HEADER=YES .... Component header, consisting of a component ID, is used. The component ID is indented two spaces. (Default)

COMPS-HEADER=NO ..... The component ID is not displayed.

COMPS-HEADER="string" ..... The specified string of up to 20 characters is used as the component header. The TFF variable @COMPS (component ID) can be used within this string. Can be truncated in the stream summary but displays in full in the stream table.

Applies to both DISPLAY and PROP sentences.

SUBSTREAM-HEADER.... Substream header

SUBSTREAM-HEADER=YES ..... Substream header of the form "Substream: ssid" is used. (Default)

SUBSTREAM-HEADER=NO ..... Substream header is not displayed.

SUBSTREAM-HEADER="string" The specified string of up to 20 characters is used as the substream header. The TFF variable @SUBSTREAM (substream ID) can be used within this string. Can be truncated in the stream summary but displays in full in the stream table.

Applies to both DISPLAY and PROP sentences.

PB-HEADER ..... Phase-Basis header

PB-HEADER=YES ..... Phase-Basis header of the form "basis phase" is used. (Default)

PB-HEADER=NO ..... Phase-Basis header is not displayed.

PB-HEADER="string" ..... The specified string of up to 20 characters is used as the Phase-Basis header. The TFF variables @BASIS (basis) and @PHASE (phase) can be used within this string. Can be truncated in the stream summary but displays in full in the stream table.

Applies to both DISPLAY and PROP sentences.

TEMP-HEADER..... Temperature header. Used when the property is calculated at a specified temperature.

TEMP-HEADER=YES ..... Temperature header of the form **Temperature value**" is used. (Default)

TEMP-HEADER=NO ..... No temperature header is displayed.

TEMP-HEADER="string" . The specified string of up to 20 characters is used as the temperature header. You can use the TFF variable @TEMP (temperature) within this string. Can be truncated in the stream summary but displays in full in the stream table.

Applies to both DISPLAY and PROP sentences

PRES-HEADER..... Pressure header. Used when the property is calculated at a specified pressure.

PRES-HEADER=YES..... Pressure header of the form"**Pressure value**" is used. (Default)

PRES-HEADER=NO ..... No pressure header is displayed.

PRES-HEADER="string" . The specified string of up to 20 characters is used as the pressure header. You can use the TFF variable @PRES (pressure) within this string. Can be truncated in the stream summary but displays in full in the stream table.

Applies to both DISPLAY and PROP sentences.

LVPCT-HEADER..... Liquid volume percent header. Used with liquid volume percent dependent properties only.

LVPCT-HEADER=YES..... Liquid volume percent header, consisting of a liquid volume percent, is used. (Default)

LVPCT-HEADER=NO ..... Liquid volume percent is not displayed.

LVPCT-HEADER="string" ..... The specified string of up to 20 characters is used as the liquid volume percent header. You can use the TFF variable @LVPCT (liquid volume percent) within this string. Can be truncated in the stream summary but displays in full in the stream table.

Applies to both DISPLAY and PROP sentences.

COMP-ATTR-HEADER.....Component attribute header

COMP-ATTR-HEADER=YES ..... Component attribute header of the form "cid cattr-id" is used. (Default)

COMP-ATTR-HEADER=NO ..... Component attribute header is not displayed.

COMP-ATTR-HEADER="string" ..... The specified string of up to 20 characters is used as the component attribute header. You can use the TFF variable @COMPS (component) and @COMP-ATTR (component attribute) within this string.

Applies to both DISPLAY and PROP sentences.

SUBS-ATTR-HEADER.....Substream attribute header

SUBS-ATTR-HEADER=YES ..... Substream attribute header of the form "*ssid sattr-id*" is used. (Default)

SUBS-ATTR-HEADER=NO ..... Substream attribute header is not displayed.

SUBS-ATTR-HEADER="string" The specified string of up to 20 characters is used as the substream attribute header. You can use the TFF variable @SUBS-ATTR (substream attribute) within this string.

Applies to both DISPLAY and PROP sentences.

COMP-ATTR-ELEM.....Component attribute element to be displayed. For example, component attribute SULFANAL has three elements: PYRITIC, SULFATE, and ORGANIC.

COMP-ATTR-ELEM=cattr-elem-list List of component attribute elements

COMP-ATTR-ELEM=ALL .All elements (Default)

Applies to both DISPLAY and PROP sentences.

SUBS-ATTR-ELEM.....Particle size distribution interval number to be displayed.

SUBS-ATTR-ELEM=sattr-elem-list ..... List of particle size distribution intervals

SUBS-ATTR-ELEM=ALL ..All intervals (Default)

Applies to both DISPLAY and PROP sentences.

Header Sentence Order in the Stream Table

The order of the header sentences (TITLE, STREAM-ID-LABEL, SOURCE-LABEL, DEST-LABEL, PHASE-LABEL) in your TFF indicates the order of header information in the stream table. The order of these sentences has no effect on the stream summary form. You must specify YES or supply your own label to display a header on the stream table.

You can specify up to 20 characters for STREAM-ID-LABEL, SOURCE-LABEL, DEST-LABEL, and PHASE-LABEL. Your specified label does not appear on the stream summary form.

Formats for Numbers

There are three conversion formats (%-*xx.yy*e, %-*xx.yy*f, %-*xx.yy*g). The conversion format variables are:

Variable	Explanation
%	Percent character. Lead character for format specification.
–	Optional minus sign, which left-justifies the number. Without the minus sign, the number is right-justified.
xx	A digit string specifying a minimum field length for the converted number. The number takes at least this much space to print, and more if necessary.
yy	A digit string specifying the precision, (that is, the number of digits) to be printed to the right of the decimal point.
e	Number is converted to the form [-]a.bbbbbbbe[+]cc. Length of b is specified by yy (Default is 6). Use uppercase E in the format specification for uppercase E in the printed numbers.
f	Number is converted to the form [-]aaa.bbbbbb. Length of b is specified by yy (Default is 6).
g	The shorter of %e or %f is used. Use uppercase G in the format specification for uppercase G in the printed numbers.

The recommended format is %10.2f. This format prints values with two digits to the right of the decimal, if there is room. If the number is greater than 9,999,999, Aspen Plus eliminates the fractional digits, then spills over the field range to the left.

Other common formats used in stream tables are:

Stream table format	Prints
%10.0f	Whole numbers, with no decimal digits or exponents
%10.nf	Numbers without exponents and with n digits to the right of the decimal point, if there is room. Decimal points line up, unless decimal digits have been eliminated in some numbers.
%10.nE	Numbers in exponential notation, with n+1 significant digits

The f format is most common in stream tables. You can use the SCALE option, or "large units" (for example, MMBTU/HR instead of BTU/HR). This option reduces the size of the value printed, so it fits in the table with the f format specification.

Any number forced to display as zero with the specified f format is displayed as "< *number*", where *number* is the smallest number that can be displayed by that format. For example, the number 0.002 displayed under the %10.2f format is < 0.01.

The NORMALIZE Option

The NORMALIZE option is used with the component flows or fraction properties, as shown in the following table:

If the component property is	Then it is normalized to
Mole flow (MOLEFLOW)	Total mole flow (MOLEFLMX) of the same substream
Mass flow (MASSFLOW)	Total mass flow (MASSFLMX) of the same substream
Standard vapor volume (VVSTD)	Total standard vapor volume (VVSTDMX) of the same substream
Standard liquid volume (VLSTD)	Total standard liquid volume (VLSTDMX) of the same substream
Mole fraction (MOLEFRAC)	1
Mass fraction (MASSFRAC)	1
Standard vapor volume fraction (VVSTDFR)	1
Standard liquid volume fraction (VLSTDFR)	1

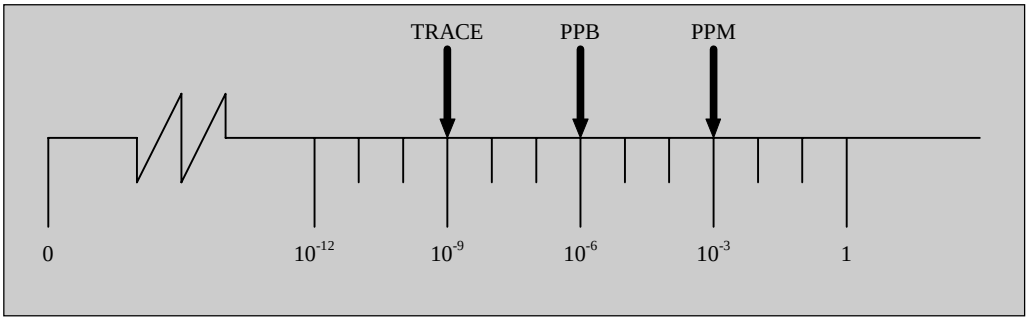
The component property displayed is forced to add up to exactly the normalization value. For example, suppose you display mass fractions with two digits (such as %10.2f) and NORMALIZE = YES. Aspen Plus adjusts the new fractions to sum to exactly 1.00, even if the value of each mass fraction rounded to 2 digits adds up to 0.99.

PPM, PPB, and TRACE Options

The options FORMAT, PPM, PPB, and TRACE are related when you specify a format on the flow or fraction properties. For example, assume you have the following specification:

PROP MOLEFLOW FORMAT="%10.3f" PPM=1e-3 PPB=1e-6 TRACE=1e-9

This specification is shown in this diagram:



If MOLEFLOW is	Then the value is displayed as
Not calculated	Blank or string specified in MISSING-LABEL
0	0.0 or string specified in ZERO-LABEL
$<10^{-9}$	Blank or string specified in TRACE-LABEL
$10^{-9} \leq$ MOLEFLOW $< 10^{-6}$	1 - 999 PPB
$10^{-6} \leq$ MOLEFLOW $< 10^{-3}$	1 - 999 PPM
$< 10^{-3}$	< 0.001
$\geq 10^{-3}$	Number as converted by the %10.3f format

You should always maintain the following relationship:

TRACE < PPB < PPM < Format precision

Example of a Full TFF

The following is the system default TFF. The intent of this TFF is to mimic the Aspen Plus stream report as closely as possible.

```
;    This TFF mimics the Aspen Plus stream report and reports all
;    calculated properties.
.;
title=yes
stream-id-label=yes
source-label=yes
dest-label=yes
phase-label=yes
;
begloop substream=all
display all
prop moleflow     prop-label="Mole Flow"
prop massflow     prop-label="Mass Flow"
prop vlstd        prop-label="Liq Vol 60F"
prop molefrac     prop-label="Mole Frac"
prop massfrac     prop-label="Mass Frac"
prop vlstdfr      prop-label="LiqVolFrac60F"
prop moleflmx     prop-label="Total Flow"
prop massflmx     prop-label="Total Flow"
prop volflmx      prop-label="Total Flow"
prop vlstdmx      prop-label="Liq Vol 60F"
prop temp         prop-label="Temperature"
prop pres         prop-label="Pressure"
prop vfrac        prop-label="Vapor Frac"
prop lfrac        prop-label="Liquid Frac"
prop sfrac        prop-label="Solid Frac"
prop hmx          prop-label="Enthalpy"
prop smx          prop-label="Entropy"
prop rhomx        prop-label="Density"
prop mwmx         prop-label="Average MW"
;
; batch properties follow
;
; first, component-dependent properties
;
prop cmass_time   prop-label = "Mass Flow"
prop cmole_time   prop-label = "Mole Flow"
prop cvol_time    prop-label = "Vol Flow"
prop cmass_cycle  prop-label = "Mass/Cycle"
prop cmole_cycle  prop-label = "Mole/Cycle"
prop cvol_cycle   prop-label = "Vol/Cycle"
;
; overall stream properties
;
prop cycle_time   prop-label = "Cycle Time"
prop oper_time    prop-label = "Operat Time"
prop ntrain       prop-label = "No. Trains"
prop mass_time    prop-label = "Mass Flow"
prop mole_time    prop-label = "Mole Flow"
prop vol_time     prop-label = "Vol Flow"
prop enth_time    prop-label = "Enthalpy"
prop mass_cycle   prop-label = "Mass/Cycle"
prop mole_cycle   prop-label = "Mole/Cycle"
prop vol_cycle    prop-label = "Vol/Cycle"
prop enth_cycle   prop-label = "Enthalpy/Cycle"
;
endloop
```

This is a general TFF that handles any stream class and any number of property sets. The BEGLOOP and ENDLOOP statements enclose the DISPLAY ALL keyword, which instructs Aspen Plus to loop through all substreams and display all properties found in each substream.

The header sentences (TITLE, STREAM-ID-LABEL, SOURCE-LABEL, DEST-LABEL, and PHASE-LABEL) are specified to take on the default values. The order of the header sentences dictates how they are printed in the stream table. Since these sentences are specified before any DISPLAY sentences, the header information is displayed above the stream values in the stream table.

Whenever Aspen Plus displays a property that is mentioned in the system default TFF, the corresponding property label is used. For example, when Aspen Plus displays density, the label "Density" is used. The default label is used for any property without a specified label. For example, the label RHOMX is used if PROP-LABEL="Density" is not specified.

All six combinations of component flow or fraction (mole, mass, standard volume) are specified to anticipate your specifications. If Aspen Plus cannot find the property you specify, nothing appears.

Example of Customizing a TFF for Generating a Stream Table

Create the following TFF file to customize your stream table:

```
title="Ethylene plant separation train - Section 105"
stream-id-label="Streams"
display  only   format="%10.2f"  substream-header=no
prop molefrac   prop-header="Comp mole fraction"   &
                comps-header="  @comps"  mw=yes mw-bp-format="%6.1f"
&
                normalize=yes  ppm=1e-3  trace=1e-7  trace-label="---
"
text " "
prop moleflmx   prop-label="Total Mole Flow"  &
                units="LBMOL/DAY"   units-label="Lbmol/Day"  &
                scale=1e3 scale-label="M"
prop massflmx   prop-label="Total Mass Flow"  &
                units="LB/DAY"  units-label="Lb/Day"  &
                scale=1e3 scale-label="M"
text " "
prop temp       prop-label="T" units="F" units-label="Deg F"
prop pres       prop-label="P" units="PSI" units-label="Psi"
prop rhomx      prop-label="Density"   units="LB/CUFT"  &
                units-label="Lb/Cuft"
```

Notice the following in the stream table:

- The stream ID label is customized. The source and destination labels are not printed.
- The DISPLAY ONLY sentence limits the display to the following properties: component mole fractions, total mole flow, total mass flow, temperature, pressure, and mass density. The format is 2-decimal-place precision.
- For the mole fraction section, the component IDs are indented two spaces. The molecular weights are displayed next to the component IDs with 1-decimal-place precision. The mole fractions are normalized to 1.
- The possible mole fraction values are displayed according to FORMAT, TRACE, PPM, and TRACE-LABEL specifications, as described in the following table:

If MOLEFRAC is	Then value is displayed as
Not calculated	Blank
0	0.0
$< 10^{-7}$	—
$10^{-7} \leq$ MOLEFRAC $\ll 10^{-6}$	< 1 PPM
$10^{-6} \leq$ MOLEFRAC $\ll 10^{-3}$	1 - 999 PPM
$10^{-3} \leq$ MOLEFRAC $\ll 10^{-2}$	< 0.01
$\geq 10^{-2}$	Number as converted by the %10.2f format

- Total mole flow is requested with units of "LBMOL/DAY." Total massflow is requested with units of "LB/DAY."
- Two blank rows are inserted for cosmetic purposes.
- Three properties (temperature, pressure, and mass density) are requested. The property labels for these properties are customized.

❖ ❖ ❖ ❖

37 Working with Other Windows Programs

The Aspen Plus Windows user interface is built using Microsoft's OLE Automation (ActiveX™) technology. This technology enables you to transfer data easily to and from other Windows® programs. It enables you to access simulation data and methods through an Automation client, such as Visual Basic® (VB).

This chapter describes how to use the Windows interoperability features. Topics include:

- About copying, pasting and OLE
- Copying and pasting simulation data
- Copying and pasting plots
- Creating active links between Aspen Plus and other Windows applications
- Using embedded objects in your flowsheet

Chapter 38 contains more information on the Aspen Plus OLE Automation server and how to access it through Visual Basic.

About Copying, Pasting, and OLE

Because Aspen Plus Version 10 is a true Windows application, you can take advantage of full Windows interoperability and object linking and embedding (OLE). You can make your simulation work more productive by creating active links between input/output fields in Aspen Plus and other applications such as Word® and Excel®.

For example, simulation results such as column profiles and stream results can be pasted into a spreadsheet for further analysis, into a word processor for reporting and documentation, into a design program, or into a database for case storage and management.

Live data links can be established that update these applications as the process model is changed to automatically propagate results of engineering changes. The benefits to you are quick and error-free data transfer and consistent engineering results throughout the engineering work process.

The subsequent sections of this chapter provide details about cutting, pasting, and creating active links using OLE.

Copying and Pasting Simulation Data

In Aspen Plus, data contained in the fields of input and result forms can be copied and pasted using the standard Copy and Paste commands on the Edit menu. For example, you can copy information from a field or group of fields in Aspen Plus, and then paste it into:

- Another location within the same Aspen Plus simulation
- Another Aspen Plus simulation
- Any other Windows application such as Word, Excel, or Access®

Copying Data

To copy information in Aspen Plus using the Copy command:

1. Select (or highlight) the information you wish to copy.

   To select an individual field of data, simply click the mouse in the field.

   To select multiple fields of data, hold down the Ctrl key while clicking the mouse on multiple fields.

   When copying values from a table, you can:

   - Click-and-drag the mouse over a desired range of results
   - Select an entire column of data by clicking the column heading
   - Select an entire row of data by clicking the row selector button (on the left of the row of data)
   - Select the entire table by clicking the button on the top left corner of the table

2. From the Edit menu, click Copy, or on the keyboard, press Ctrl + C.

   The selected values are now contained in the Windows paste buffer, and can be pasted into Aspen Plus, or another Windows application.

Note: When selecting data to be copied from a field in Aspen Plus, the entire field of information is copied, not just a selected portion of the field. For example, if a field contains as its value the number “1234.567”, you cannot use the mouse to highlight a portion of the value (such as “123”) for copying.

The Copy command always copies the whole field, with these exceptions:

- The Setup Specifications Description sheet
- Any Fortran or Declarations input sheet
- The Comments dialog box for individual forms

Use the text box on these sheets for entering information, and to select and copy information.

Copying with Format

By default, the Copy command copies only the value (or values) of information. Use the Copy with Format command from the Edit menu to request that the label, units and basis for the values be included with the value.

To copy information in Aspen Plus using the Copy with Format command:

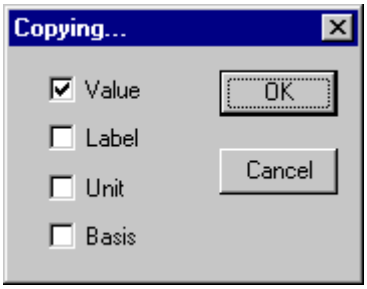
1. Select (or highlight) the information you wish to copy.

   To select an individual field of data, click the mouse in the field.

   To select multiple fields of data, hold down the Ctrl key while clicking the mouse on multiple fields.

   Tip: When copying values from a table, you can click-and-drag the mouse over a desired range of results, or you can select an entire column or row of data by clicking the column heading or row selector button.

2. From the Edit menu, click Copy with Format.
3. On the Copying dialog box that appears, click the check boxes representing the type(s) of information that you want to be included in the copy buffer.



4. Click OK.

   The selected information is now contained in the Windows paste buffer, and can be pasted into Aspen Plus, or another Windows application.

Note: You can change the default formats included with the standard Copy command, by selecting options in the Copy Buffer Format frame of the General sheet on the Tools Options dialog box. See Chapter 16 for more details on the Tools Options dialog box.

Pasting

To paste information in Aspen Plus using the Paste command:

1. First, ensure that the paste buffer contains information that has been copied from Aspen Plus, or another Windows application.
2. Click the mouse in the input field where you wish to paste the information. For multiple fields of information, click in the upper-left most field.
3. From the Edit menu, click Paste or on the keyboard, press Ctrl + V.
4. If prompted with a message asking if you want to extend the grid, click Yes. Aspen Plus needs to extend the grid if you are pasting more rows or columns of data than are currently displayed.

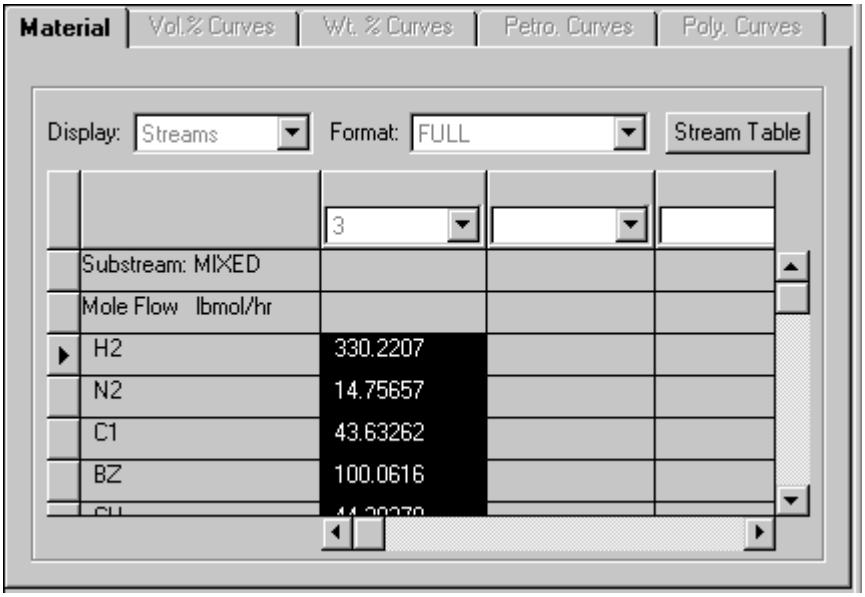
   The information contained in the paste buffer will now appear in the field, or group of fields you selected with the cursor. This information remains in the paste buffer, and can be pasted into additional locations by repeating steps 2 through 4.

Note The Paste command has automatic filtering which prevents the pasting of inconsistent or inappropriate information. For example, you cannot paste a real value into an integer input field.

Example of Cutting and Pasting Within Aspen Plus

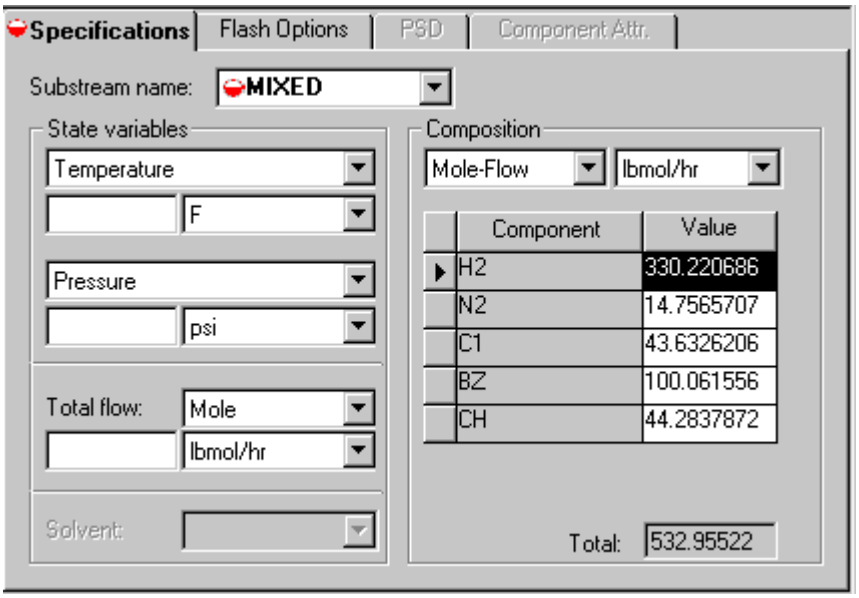
In this example, stream results are pasted into stream input fields. This is a common task when you want to save final results as initial estimates for tear streams.

1. Open stream results. To do this, click the tear stream to select it, then click it with the right mouse button. On the popup menu that appears, click Results.
2. In the Data Browser, click the left mouse button on the molar flowrate of the first component in the list, and drag the mouse down to select all the values for component molar flowrates.



Tip If all the component flowrates are not displayed, you can expand the Data Browser window to display more components. Or you can select the component flowrates by holding down the Ctrl key and clicking with the mouse.

3. From the Edit menu, click Copy.
4. Using the Data Browser, open the Streams Input Specifications sheet for the tear stream.
5. In the Composition frame, click in the first cell in the Value column.
6. From the Edit menu, click Paste.



The molar flowrates from the stream results have been copied into the stream input specifications. You can now enter two state variables (you could also copy these values if you wish) to complete the initial estimates for this tear stream.

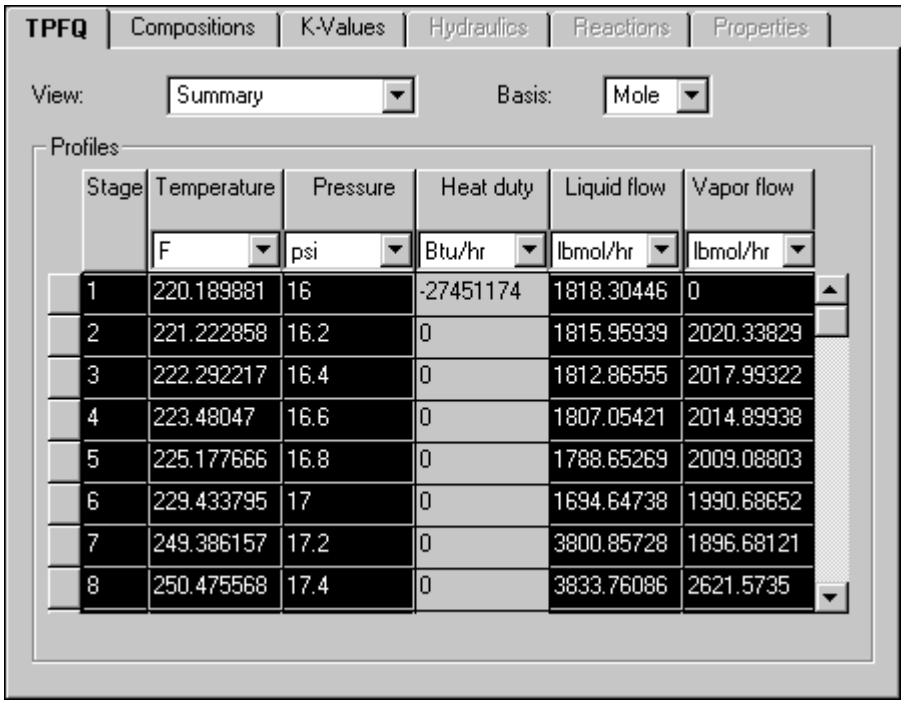
Example of Pasting Aspen Plus Results Into Other Applications

This example shows the steps necessary to paste column profile results from a RadFrac block in Aspen Plus into an Excel spreadsheet.

1. Open the column profile results. To do this, in the Process Flowsheet window, click the RadFrac column to select it, then click it with the right mouse button. From the menu that appears, click Results.
2. In the left pane of the Data Browser window, click the Profiles results form.
3. On the Profiles result form, click and drag over the results you wish to copy.

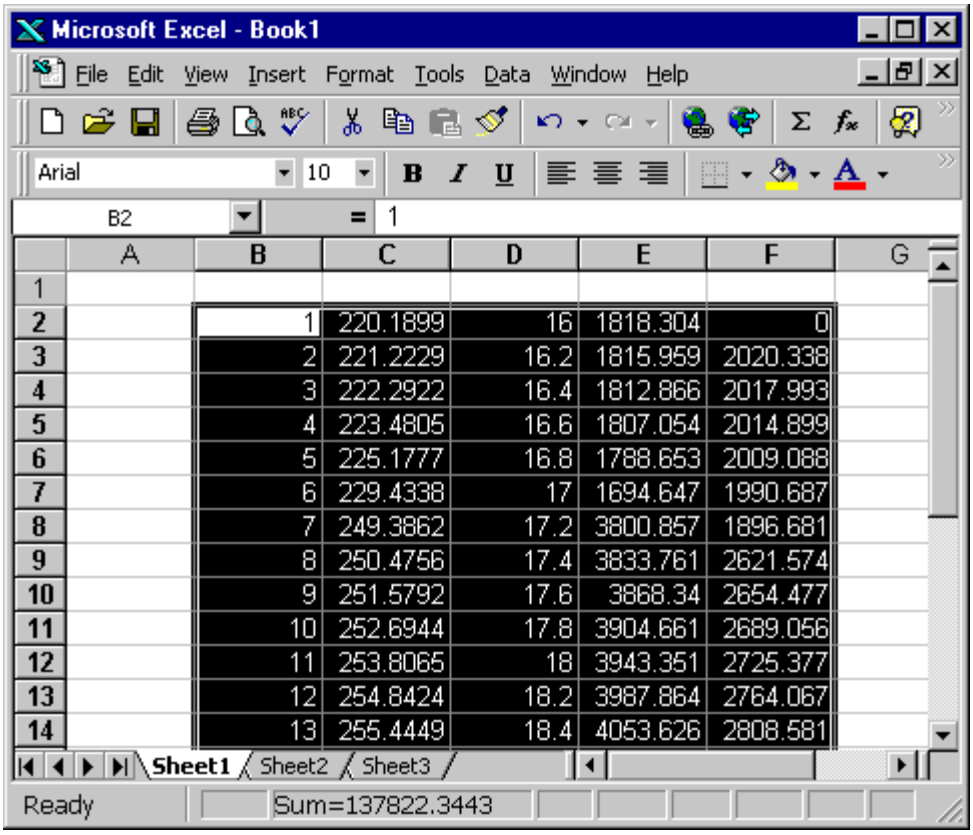
   – or –

   Hold down the Ctrl key while you click the column headings for the data you wish to copy.



Stage	Temperature	Pressure	Heat duty	Liquid flow	Vapor flow
	F	psi	Btu/hr	lbmol/hr	lbmol/hr
1	220.189881	16	-27451174	1818.30446	0
2	221.222858	16.2	0	1815.95939	2020.33829
3	222.292217	16.4	0	1812.86555	2017.99322
4	223.48047	16.6	0	1807.05421	2014.89938
5	225.177666	16.8	0	1788.65269	2009.08803
6	229.433795	17	0	1694.64738	1990.68652
7	249.386157	17.2	0	3800.85728	1896.68121
8	250.475568	17.4	0	3833.76086	2621.5735

4. From the Edit menu, click Copy.
5. Open a spreadsheet in Excel.
6. Select a cell in the Excel spreadsheet where you want to paste the information.
7. From the Edit menu in Excel, click Paste.



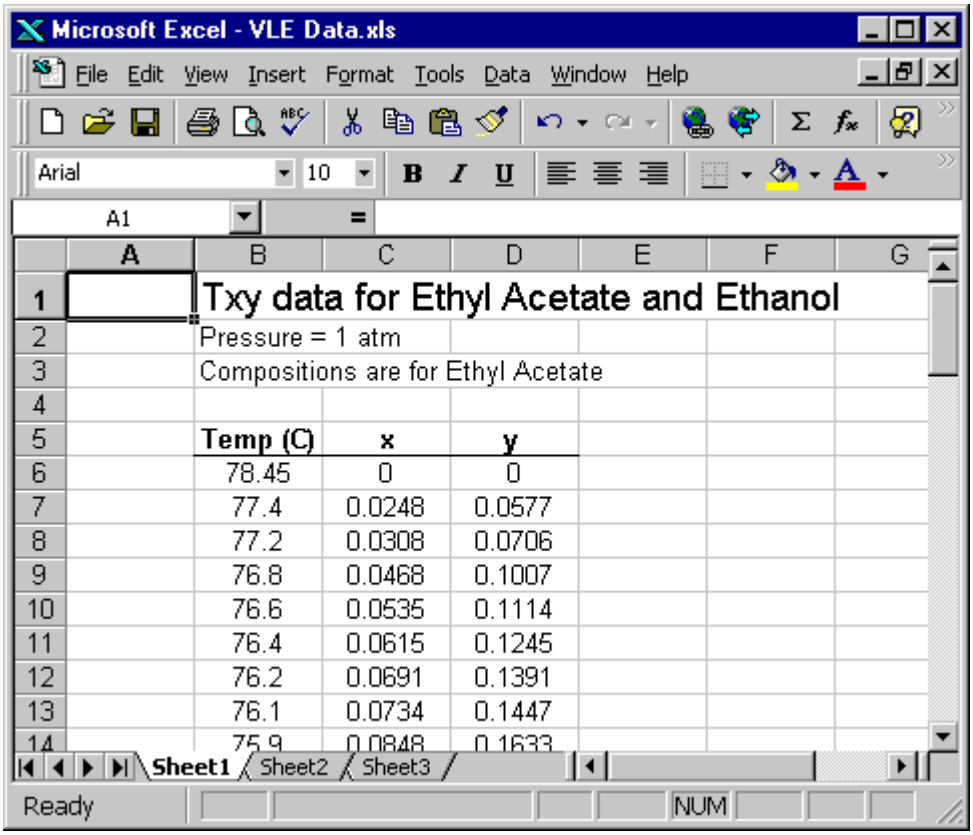
	A	B	C	D	E	F	G
1							
2		1	220.1899	16	1818.304	0	
3		2	221.2229	16.2	1815.959	2020.338	
4		3	222.2922	16.4	1812.866	2017.993	
5		4	223.4805	16.6	1807.054	2014.899	
6		5	225.1777	16.8	1788.653	2009.088	
7		6	229.4338	17	1694.647	1990.687	
8		7	249.3862	17.2	3800.857	1896.681	
9		8	250.4756	17.4	3833.761	2621.574	
10		9	251.5792	17.6	3868.34	2654.477	
11		10	252.6944	17.8	3904.661	2689.056	
12		11	253.8065	18	3943.351	2725.377	
13		12	254.8424	18.2	3987.864	2764.067	
14		13	255.4449	18.4	4053.626	2808.581	

The copied RadFrac results profile has been pasted into the spreadsheet, where it can be manipulated, reformatted, combined with additional data, and plotted using the features of Excel. This same data could also be pasted into other applications such as a table in Word, or a database in Access.

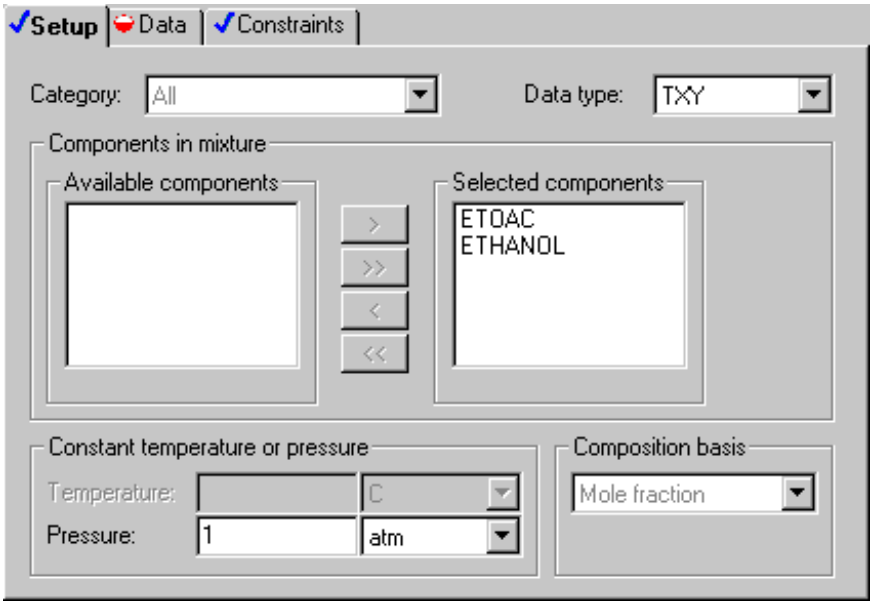
Example of Pasting Data From Another Application Into Aspen Plus

In this example, atmospheric Txy data for ethyl acetate and ethanol will be copied from an Excel spreadsheet and pasted into a Properties Data form in Aspen Plus.

The Excel data is shown below:



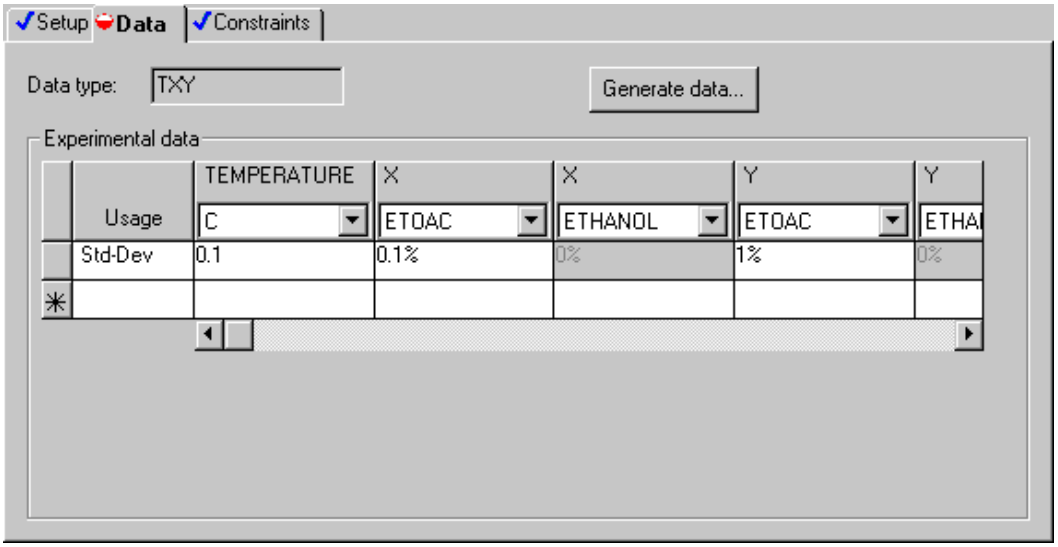
1. In the Aspen Plus simulation, create a properties data set of an appropriate type to input the data. In this case, open or create a mixture Txy data set (from the Properties Data Object Manager) for the components ethyl acetate and ethanol, at a pressure of 1 atmosphere. For more information on creating property data sets, see Chapter 31.



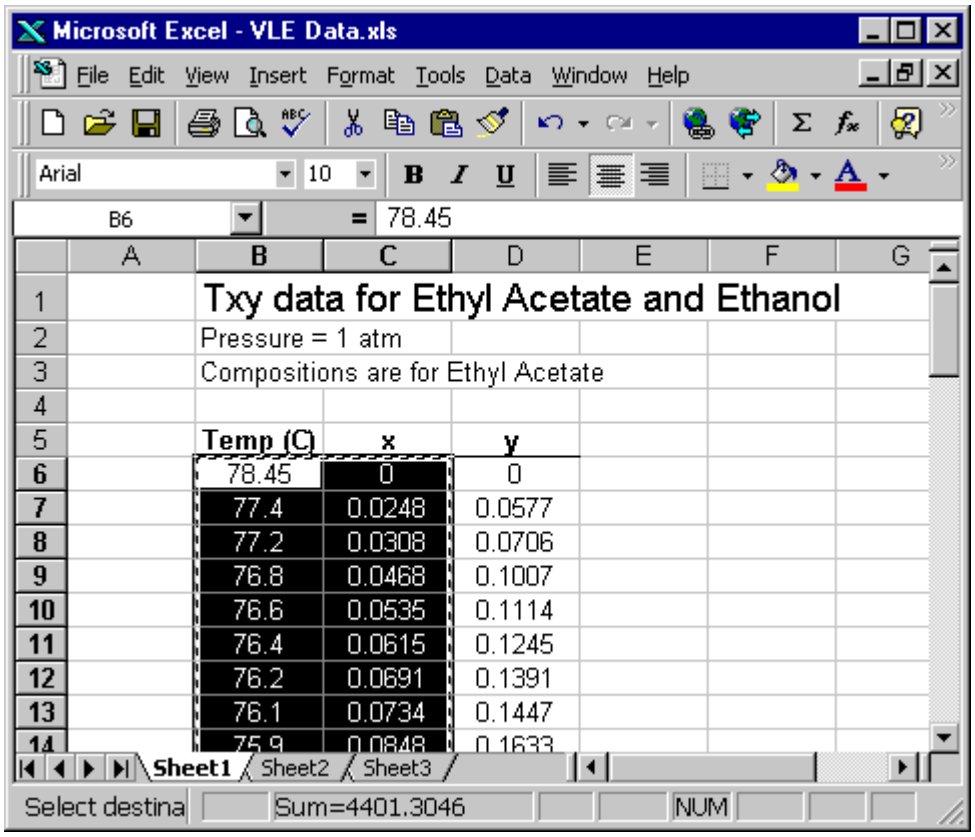
2. Open the Properties Data mixture form for the newly created data set, and examine the format for the columns of data. Modify units and standard deviations if necessary.

   Notice that there are columns for the compositions of ethanol, as well as ethyl acetate. The composition for the second component need not be entered, as it will be calculated as the difference between 1 and the composition of the first component. This means that you must use two copy and paste operations to transfer the data from Excel to Aspen Plus:

   - Firstly, copy the Temperature and X columns
   - Then copy the Y column

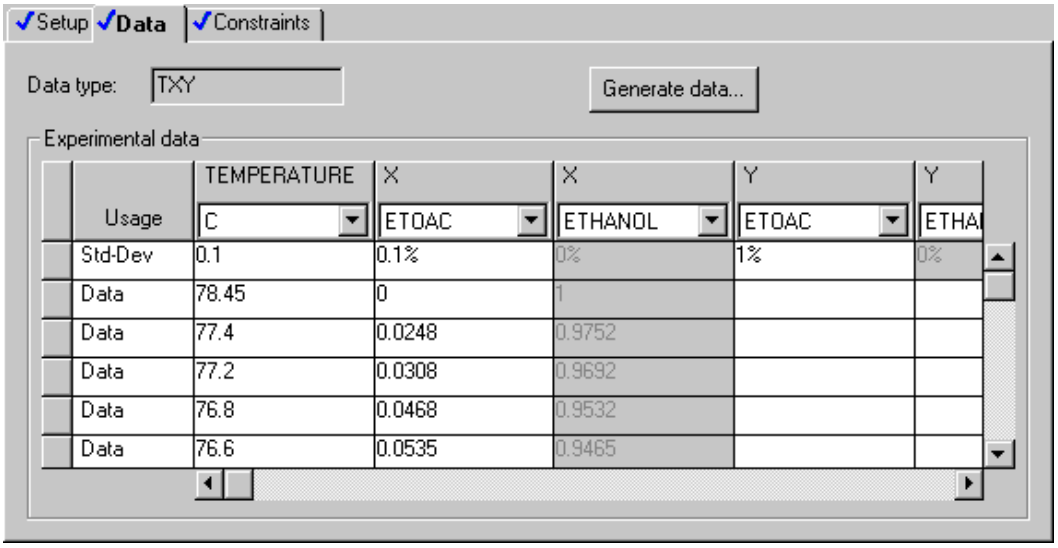


3. Open the Excel spreadsheet containing the data.
4. Click and drag to select the data to be copied. For the first copy operation, select the temperature and liquid composition data.



5. From the Edit menu in Excel, click Copy.
6. In Aspen Plus, on the Data sheet for the newly created data set, select the first empty cell in the Temperature column.
7. From the Aspen Plus Edit menu, click Paste.
8. In the Paste dialog box, click Yes to extend the data grid.

   The temperature and liquid composition data is transferred into the Data sheet.



9. Return to the Excel spreadsheet, and select the vapor composition data, by clicking and dragging the mouse.
10. From the Edit menu in Excel, click Copy.
11. In Aspen Plus, on the Data sheet, select the first empty cell in the Y column for ethyl acetate.
12. From the Aspen Plus Edit menu, click Paste.

    The vapor composition data is transferred into the Data sheet.

    You can now use this data set to estimate or regress property parameters in Aspen Plus. For more information on Property Estimation or Property Regression, see Chapters 30 and 31 respectively.

Copying and Pasting Plots and Other Images

After generating plots in Aspen Plus, you can copy the plots and paste them into the process flowsheet or into other Windows applications as images. Plotted results can be of column profiles, sensitivity analyses, batch profiles, property analyses, and so on. You can also copy images created in other Windows applications, such as graphs created in Excel, and paste them into the Aspen Plus process flowsheet.

To copy a plot in Aspen Plus:

1. Generate the desired plot and format the appearance of the plot as you want it to appear when pasted. For details on creating and formatting plots, see Chapter 13.
2. Select the plot in the Aspen Plus main window.
3. From the Edit menu, click Copy.

   The plot is copied to the paste buffer.

   Use Paste to paste the plot into the process flowsheet, or into other Windows applications.

To paste a plot into the process flowsheet:

1. Ensure that the paste buffer contains the desired plot or other image.
2. Click in an empty part of the process flowsheet.
3. From the Edit menu, click Paste.

   The image appears as an icon in the process flowsheet.

   You can move or resize the image, like any object in the flowsheet drawing. If the image is a plot, you can also change its formatting. To do this, click the plot with the right mouse button. From the menu that appears, click Properties. For more information on formatting plots, see Chapter 13.

Attaching Plots or Images to Flowsheet Blocks

When a plot or image has been pasted onto the process flowsheet, you can attach (or associate) the image to a block on the flowsheet. Attaching an image to a flowsheet block ensures that when the block is moved, the image will maintain is location with respect to the block.

For example, if you paste a plot of column profiles onto the flowsheet, you typically want this to be displayed near the column.

To attach an image to a flowsheet block:

1. Select the image that you wish to attach.
2. Click with the right mouse button on the image, and from the menu that appears, click Attach.

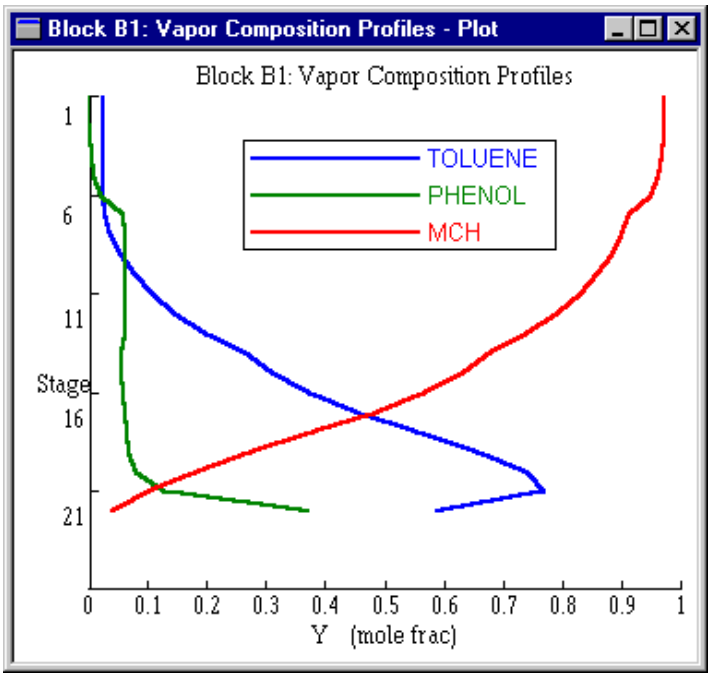
   The cursor changes to a cross-hair symbol.
3. Click the flowsheet block to which you want to attach the image.

   The image is now attached to the selected flowsheet block. If the block is later moved to another location on the flowsheet, the image will maintain its spatial arrangement with respect to the block.

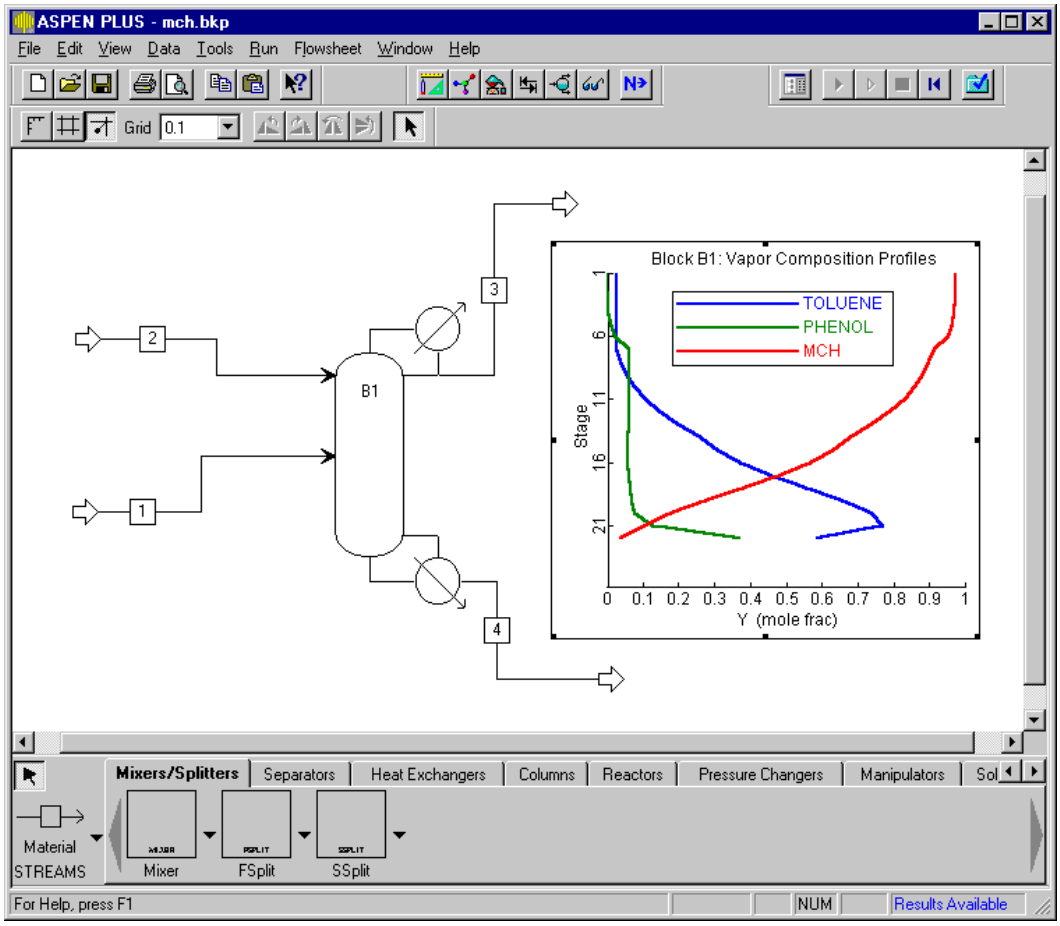
Example of Copying a Plot and Pasting it onto the Process Flowsheet

In this example, a plot of RadFrac composition profiles will be copied and pasted onto the process flowsheet.

1. First, use the Plot Wizard to generate the plot of composition profiles, and format it as you wish.



2. Select the plot and from the Edit menu, click Copy.
3. Click in an empty area of the process flowsheet.
4. From the Edit menu, click Paste.
5. Position and size the plot as needed.



Example of Pasting a Plot into Another Application

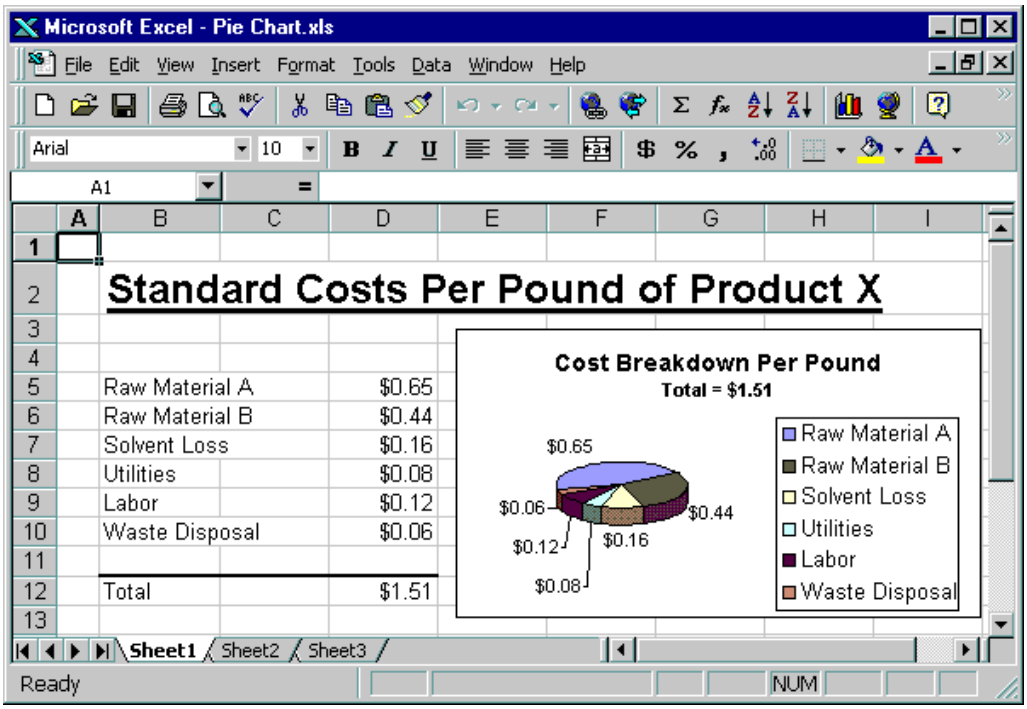
In this example, the plot copied in the previous example will be pasted into a Word document.

1. First, use the Plot Wizard to generate the plot of composition profiles, and format it as you wish. See Chapter 13 for details on generating plots.
2. Select the plot, then from the Edit menu, click Copy.
3. Start Word, and open the file in which you want to paste the plot.
4. Click in the Word document where you want to paste the plot.
5. From the Edit menu, click Paste.

Example of Pasting Images From Other Windows Applications Onto the Aspen Plus Process Flowsheet

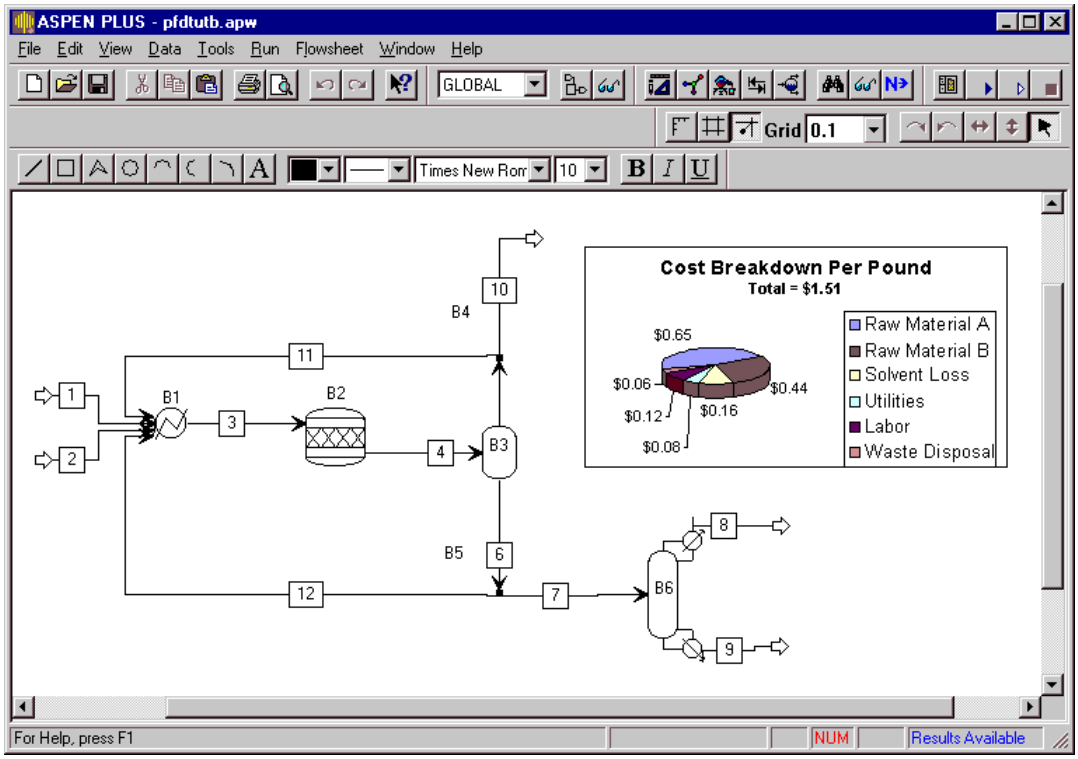
In this example, an Excel graph will be placed onto the Aspen Plus process flowsheet.

1. First, generate the desired graph or image in another application. In this case, a pie chart is created in Excel.



2. In Excel, select the graph, and from the Edit menu, click Copy.
3. Open the Aspen Plus simulation where you want to paste the Excel graph.
4. Click on an empty area of the process flowsheet.
5. From the Edit menu in Aspen Plus, click Paste.
6. Move and resize the graph, and adjust the flowsheet view as necessary.

   The Excel graph now appears on the process flowsheet.



Creating Active Links Between Aspen Plus and Other Windows Applications

When copying and pasting information, you can create active links between input or results fields in Aspen Plus and other applications such as Word and Excel. The links update these applications as the process model is modified to automatically propagate results of engineering changes.

Creating Active Links Between an Aspen Plus Result and another Windows Application

To create active links between a result in Aspen Plus and another Windows application:

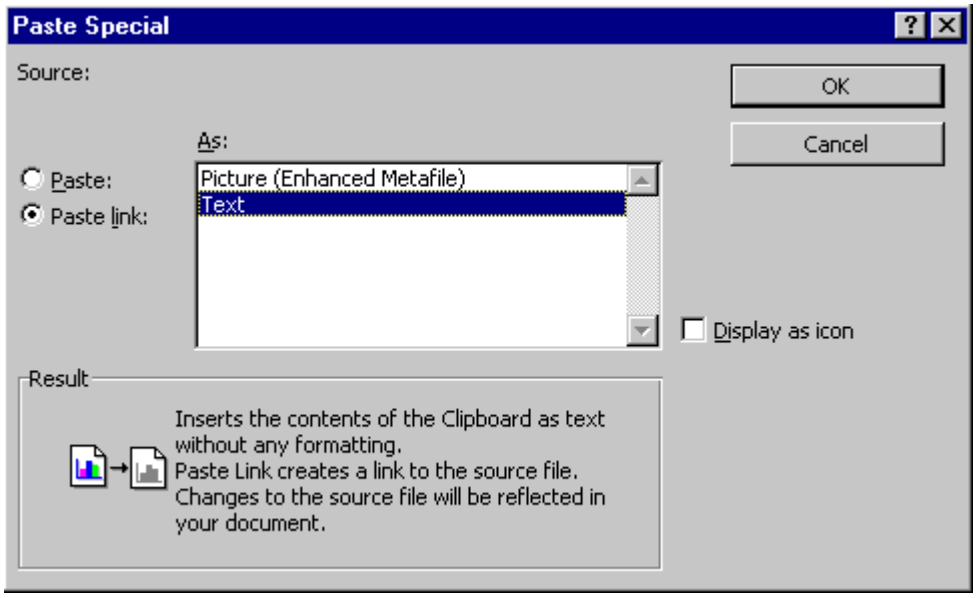
1. Make sure you have both applications open:
   - Aspen Plus open with the completed simulation and results available
   - Another Windows application open with the file where you wish to paste the active link to Aspen Plus results
2. Open the Aspen Plus results form containing the information to be linked.
3. Select the desired results.

   To select an individual field of data, simply click in the field.

   To select multiple fields of data, hold down the Ctrl key while clicking the mouse on multiple fields.

   When copying values from a table, you can:

   - Click-and-drag the mouse over a desired range of results
   - Select an entire column of data by clicking the column heading
   - Select an entire row of data by clicking the row selector button
   - Select the entire table of data by clicking the button on the top left corner of the table
4. From the Edit menu, click Copy (or Copy with Format). If you choose Copy with Format, in the Copying dialog box, check the items you want included with the value (Label, Units or Basis), and click OK.
5. Go to the appropriate location in another Windows application, where you wish to paste the active link.
6. From the Edit menu in the other application, choose Paste Special.
7. In the Paste Special dialog box, click the Paste Link radio button and make sure you are pasting as text by selecting Text in the As box:.
8. Click OK to close the Paste Special dialog box.



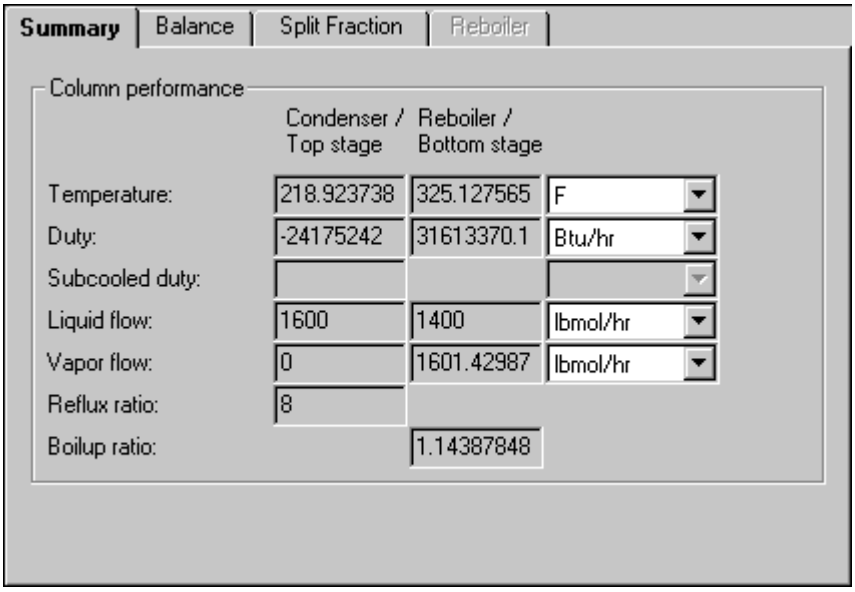
Now an active link has been established between Aspen Plus (the source document) and another application.

9. When you exit, be sure you save both the Aspen Plus file and the other application file. If you do not, the link will not work when you open the files. If you save the link source file (Aspen Plus in this case) with another name, you must save the link container (other application file) **after** saving the Aspen Plus run.

Example of Creating Active Links from Aspen Plus Results into Excel

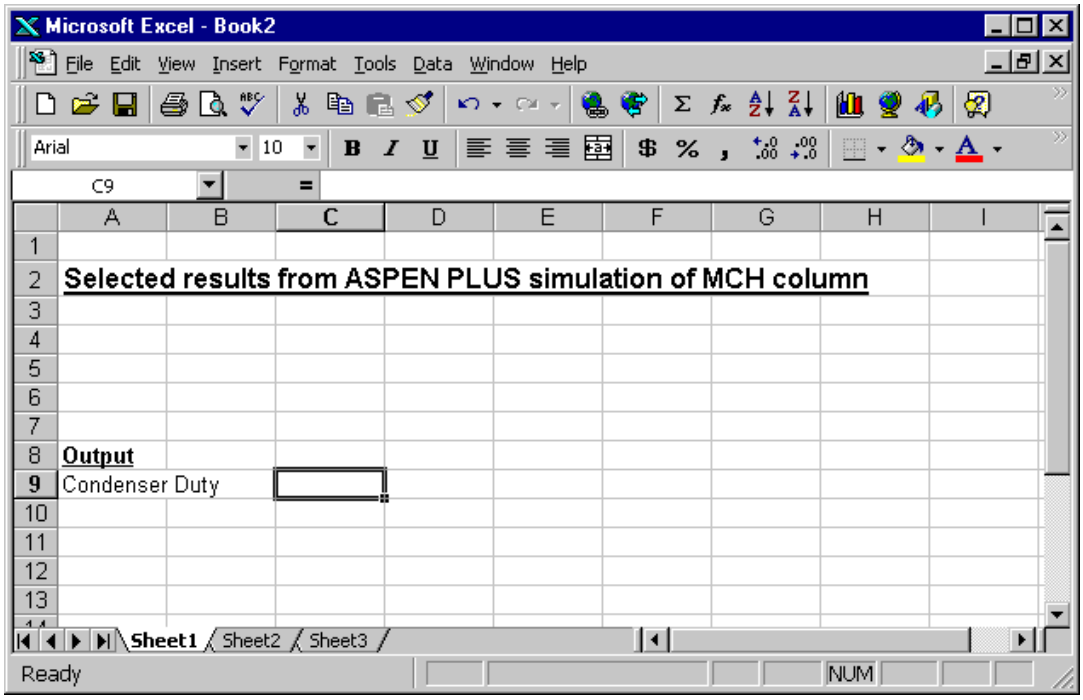
In this example, RadFrac condenser duty results will be copied with units, and pasted into an Excel spreadsheet as an active link.

1. Open the RadFrac ResultsSummary Summary sheet to view the results for condenser duty.



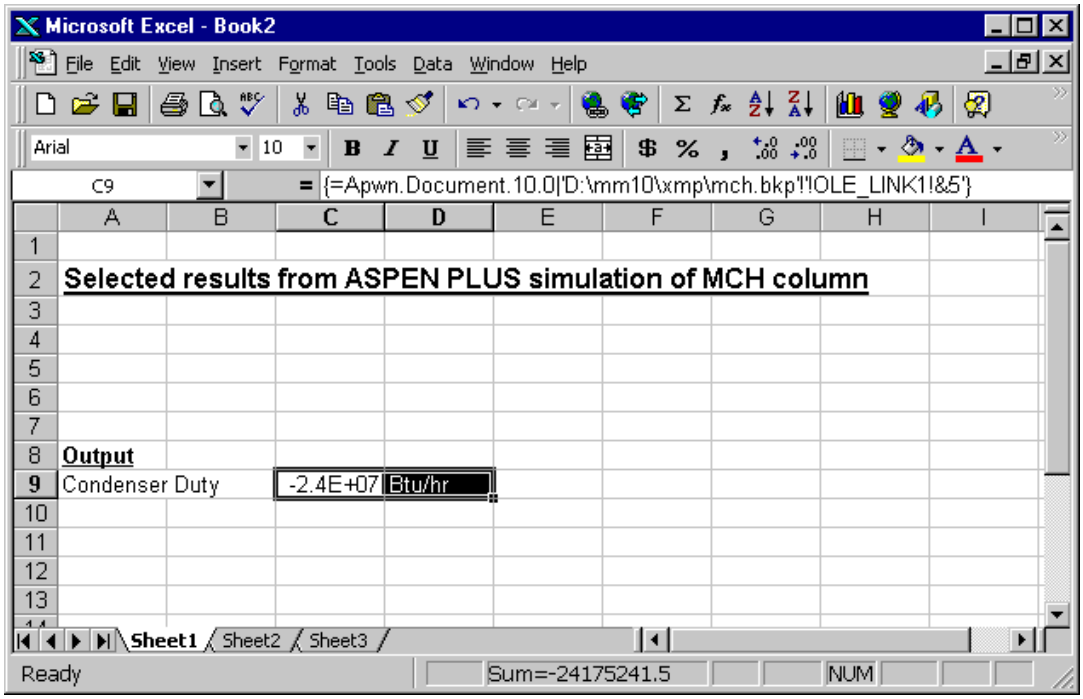
2. On the Summary sheet, click the result value for condenser duty.
3. From the Edit menu, click Copy with Format.
4. In the Copying dialog box, check the Unit checkbox, then click OK.

5. Open the Excel spreadsheet, and select the cell where you want to create the link to the Aspen Plus results for condenser duty.



6. From the Edit menu in Excel, choose Paste Special.
7. In the Paste Special dialog box, click the Paste Link radio button.
8. Select Text in the As: list, and click OK.

   The condenser duty and units are copied into the specified location.



The pasted value is an active link between Aspen Plus (the source document) and the Excel spreadsheet (the destination document.) As inputs are changed in the Aspen Plus model, and the simulation is rerun to generate new results, the active link displayed in the Excel spreadsheet will reflect the changes.

You can review the source of the link in Excel by selecting the linked cell in Excel. The source will display in the Excel Formula Bar below the toolbar.

You can view and modify the status of the link in Excel by selecting Links from the Edit menu.

Creating Active Links from a Windows Application to Aspen Plus Input Fields

In addition to creating active links from Aspen Plus to other applications, you can also create active links from other applications such as Word or Excel, to input fields within Aspen Plus simulations. This can be used to create a simple interface to your simulation models for non Aspen Plus users (e.g. operators or other engineers.)

1. Make sure you have:
   - Aspen Plus open at the completed simulation where you will add the active link
   - Another Windows application open at the source file from where you will originate the active link to an Aspen Plus input field
2. In the source file of the other application, select the information to be linked. For example, in Excel, click in the cell containing the data to be linked.
3. From the Edit menu in the other application, click Copy.
4. In Aspen Plus, open the appropriate input form, and select the field where the information will be pasted to create the active link.
5. From the Edit menu in Aspen Plus, click Paste Special.
6. In the Paste Special dialog box, click the Paste Link button and ensure you are pasting as text by selecting Text in the As list.
7. Click OK to close the Paste Special dialog box.

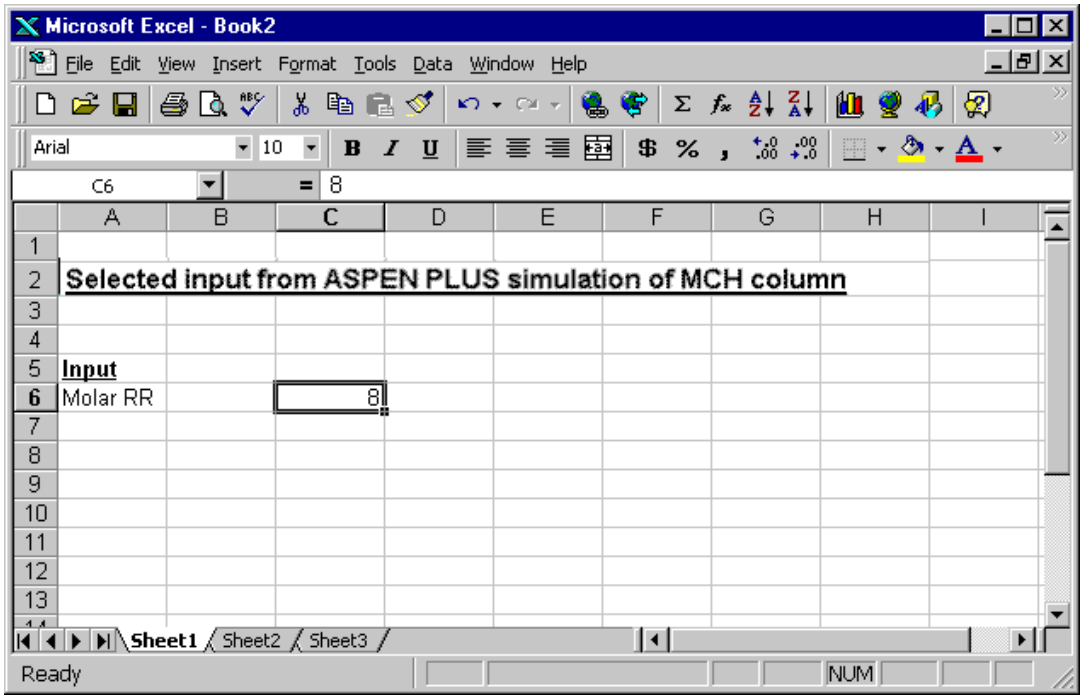
   Now an active link has been established between another application (the source document) and Aspen Plus (the destination document); if you change a value in the source document, the change will be reflected on the appropriate Aspen Plus input forms.

8. When you exit, ensure you save both the Aspen Plus file and the other application file. If you do not, the link will not work when you open the files. If you save the link source (the other application in this case) with a different file name, you must save the link container (Aspen Plus) **after** saving the other application file.

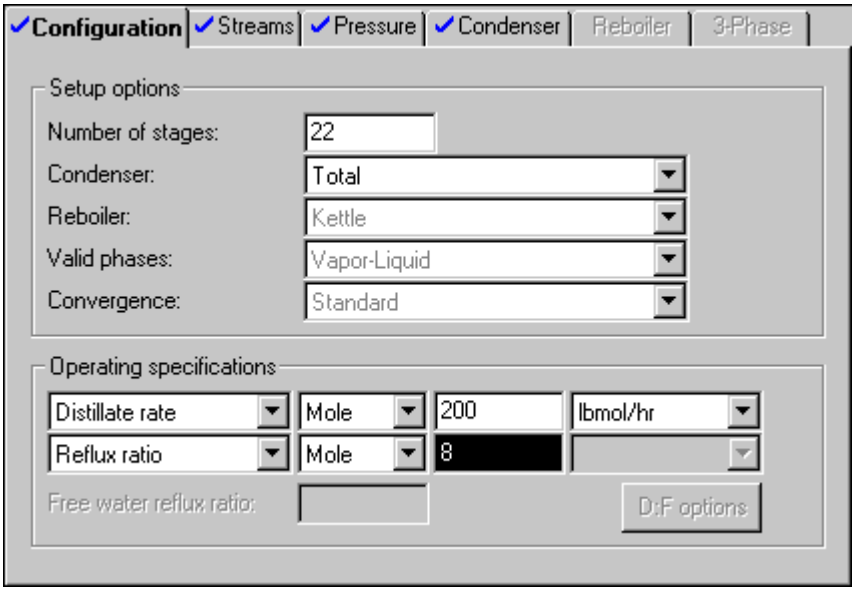
Example of Creating a Link from Excel to an Aspen Plus Input Field

In this example, an active link will be established from Excel that controls the reflux ratio of a column in Aspen Plus.

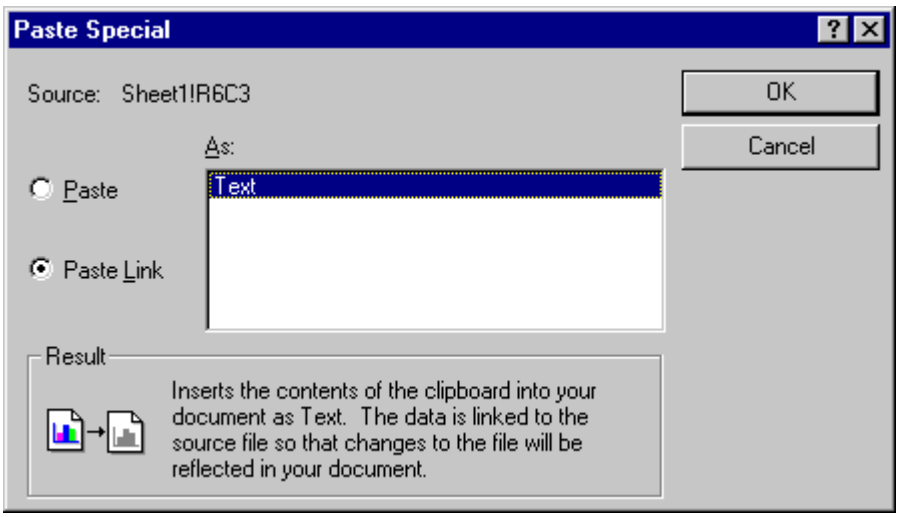
1. Open the Excel spreadsheet to display the data from where you will establish the link (the source file.)
2. Select the cell containing the information that will be linked.



3. From the Edit menu in Excel, click Copy.
4. In the Aspen Plus simulation, open the RadFrac Setup Configuration sheet, and select the field for the value of Reflux Ratio.



5. From the Edit menu in Aspen Plus, click Paste Special.
6. In the Paste Special dialog box, click the Paste Link radio button and make sure you are pasting as text by selecting Text in the As list.



7. Click OK to close the Paste Special dialog box.

   The reflux ratio displayed on the RadFrac Setup Configuration sheet is now an active link to the source cell in the Excel spreadsheet. Any changes made to the linked cell in the Excel spreadsheet will automatically be reflected in the simulation input.

To illustrate the effect of the active link established in this example:

1. Open the Excel spreadsheet, and change the reflux ratio in the linked cell from 8 to 10.

2. Open the RadFrac Setup Configuration sheet again, and note that the value for reflux ratio has been automatically changed.

Saving and Opening Files with Active Links

If you create active links between Aspen Plus and other Windows programs, you must follow a few rules to ensure that the links continue to work when you save files and open them again. You should understand the following terms:

- The *link source* is the program that is providing the data.
- The *link container* is the program into which you paste the link.

For example, if you copy data from Aspen Plus and use Paste Special to paste a link into Excel, Aspen Plus is the link source and Excel is the link container.

Saving Files with Active Links

When you save files with active links:

- Be sure to save both the link source file and the link container file. If you do not, the link will not be there when you open the files again.
- If you save the link source with a different name (for example, using Save As), you must save the link container after saving the link source. This is because the link container contains the file name of the link source.
- If you have active links in both directions between the two applications and you change the name of both files, you must do three Save operations:
  - Save the first application with a new name
  - Save the second application with a new name
  - Save the first application again

  For example, if you have links in both directions between Aspen Plus and Excel:

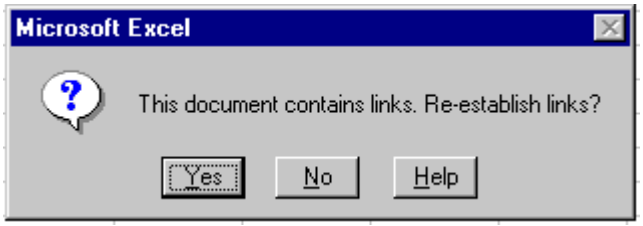
  - Use Save As in Aspen Plus to save the run as MYRUN
  - Go to Excel and use Save As
  - Return to Aspen Plus and Save

Note: Links are saved when you save in Aspen Plus Document format (.apw) or Aspen Plus Backup format (.bkp).

Opening Files with Active Links

When you open the link source file, there is nothing special that you need do.

When you open the link container file, you will usually see a dialog box asking you if you want to re-establish the links. Applications will behave differently or may show different dialog boxes.



If you:

Click	Then	And
No	The link will not be active	Any changes you make in the link source will not be reflected in the link container.
Yes	Windows will re-establish the link and open the link source application in background. That is, the application will be open and running, but there will be no visible windows for the application.	You will not see the application on the Windows taskbar. You might notice a pause as Windows activates the application.

In some cases, when the link source is running in background, you might want to make the application visible (have its windows displayed) so that you can make changes.

For example, you may be using Excel (as link source) to supply feed stream data to an Aspen Plus simulation (the link container). Normally, you can just open Aspen Plus, re-establish the links, and run the simulation. But if you want to change the feed stream data or add links to a another piece of data in the Excel spreadsheet, you need to make Excel visible.

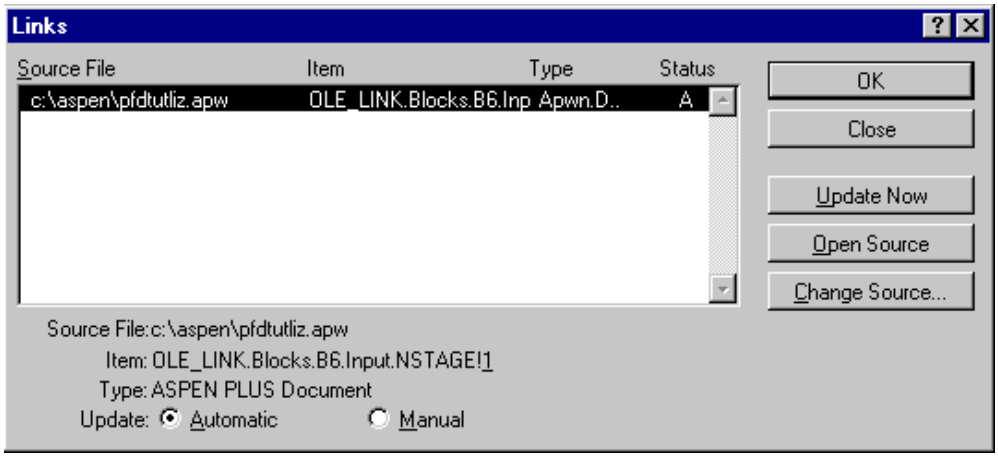
The method to make the link source application visible depends on the application:

- For some applications, for example Aspen Plus and Microsoft Word, you can open the file in the normal way using Open from the File menu or double-clicking the file in Windows Explorer.
- For other applications, like Excel, if you try to open the file in the normal way, you will receive a message that the file is locked or in use by another user. If you proceed and open the file, you are actually working on another copy of the document and links will not work properly.

Because of problems with some applications, follow this procedure to make the link source application visible:

1. From the Edit menu in the link container application, select Links.

   The Links dialog box appears.



2. In the Links dialog box, select the source file and click Open Source.

   Now the link source application is visible. The application will appear on the Windows taskbar.

Updating References in Excel

Microsoft Excel has an option which you must use to ensure that links are correctly re-established when you open files with active links. To check the option:

1. In Excel, from the Tools menu, click Options.
2. In the Options dialog box, click the Calculations tab.
3. Ensure the Update Remote References checkbox is selected.

   There is also an option to Save External Link Values. This controls the behavior of Excel when you have links but do not re-establish them when you open the file or the links become broken.

If this option is	Excel will display
Selected	The last value it had before the link was broken
Clear	An error

Using Embedded Objects in the Process Flowsheet Window

You can embed other applications as objects in the Process Flowsheet window.

For example, you can embed a Word document or an Excel spreadsheet into the Process Flowsheet window. There are two ways you can do this:

- Using Copy and Paste
- Using the Insert dialog box

Embedding an Object Using Copy and Paste

To embed an object using Copy and Paste:

1. In the source application, select the data, text, or other object you want to embed.
2. From the Edit menu of the source application, click Copy.
3. Go to Aspen Plus and make sure that the Process Flowsheet Window is the current window:

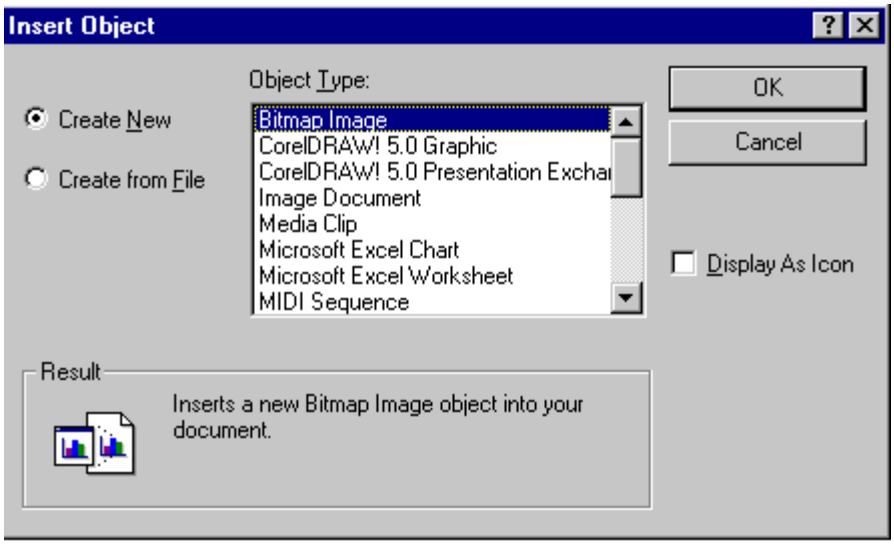
If you are using	You should
Workbook mode	Click the Process Flowsheet tab
Flowsheet as Wallpaper	Click the flowsheet in the background
Normal View	Select the Process Flowsheet window

4. In Aspen Plus, from the Edit menu, click Paste.

Embedding an Object Using the Insert Object Dialog Box

To embed an object using the Insert Object dialog box:

1. In Aspen Plus, ensure that the Process Flowsheet Window is the current window.
2. From the Edit menu, point to Insert, then New Object.



3. To embed a new object, click Create New and in the Object Type list, select the application or object.

   To embed an object from an existing file, select Create From File and specify the file.

Modifying an Embedded Object

You can modify an embedded object:

To	Do this
Edit the object using the source application within Aspen Plus	Double-click on the object. – or – 1. Click the object to select it 2. Click the right mouse button. 3. On the popup-menu, point to *Objecttype*† Object, then Edit. The menus and toolbar in Aspen Plus are replaced with those of the source application. You can edit the object. When you are done, click anywhere in the Process Flowsheet Window to exit the application.
Activate the source application to edit the object	1. Click the object to select it 2. Click the right mouse button. 3. On the popup-menu, point to *Objecttype*† Object, then Open. The source application opens in another window. You can edit the object. When you exit the application, the object is updated in Aspen Plus.
Move the object	1. Click the object to select it. The mouse pointer becomes the move shape ✥ 2. Click and hold the left mouse button while dragging the object

Continued

To	Do this
Resize the object	1. Click the object to select it. 2. Move the mouse to the edge or corner of the object. The mouse pointer becomes to the resize shape 3. Hold down the left mouse button and drag the cursor until the object is the desired size.
Attach the object to a block or stream in the flowsheet	1. Click the object to select it. 2. Click the right mouse button. 3. On the popup menu, click Attach. 4. Click the block or stream in the flowsheet. The object is now attached to the selected block or stream. If the block or stream is later moved to another location in the flowsheet, the image will maintain its spatial arrangement with respect to the block.

† *Object type will depend on the source application*

Saving a Run With an Embedded Object

Embedded objects are saved as part of a run only when you save in Aspen Plus Document format (.apw files). When you save in Backup format (.bkp files), the embedded object is not saved.

38 Using the Aspen Plus ActiveX Automation Server

This chapter describes how to use the Aspen Plus ActiveX Automation Server. The topics include:

- About the Automation server
- Viewing the properties and methods of Aspen Plus objects
- Objects exposed by the Automation server
- Using the Variable Explorer to navigate the tree structure
- Navigating the tree structure in the Automation interface
- Data values and Node attributes
- Physical quantities and Units of Measure
- Referencing non-scalar data
- Controlling a simulation problem
- Flowsheet connectivity and automation
- Members of Aspen Plus classes

This chapter assumes that you are familiar with Visual Basic and understand the concepts of object-oriented programming.

The examples in this chapter use Visual Basic 5.0 and Visual Basic for Applications (VBA) as the Automation Client. Examples are based on the pfdtut example problem which is provided with the standard Aspen Plus installation as a backup file named pfdtut.bkp. If you installed Aspen Plus in the default location, this file is in Program Files\AspenTech\APUI100\xmp.

If you installed Aspen Plus in the default location, the Visual Basic examples in this chapter are located in Program Files\AspenTech\APUI100\vbexample.

About the Automation Server

The Aspen Plus Windows user interface is an ActiveX Automation Server. The ActiveX technology (also called OLE Automation) enables an external Windows application to interact with Aspen Plus through a programming interface using a language such as Microsoft's Visual Basic. The server exposes objects through the COM object model.

With the Automation interface, you can:

- Connect both the inputs and the results of Aspen Plus simulations to other applications such as design programs or databases.
- Write your own user interface to an Aspen Plus plant model. You can use this interface to distribute your plant model to others who can run the Aspen Plus model without learning to use the Aspen Plus user interface.

Using the Automation Server

In order to use the Aspen Plus Automation Server, you must:

- Have Aspen Plus installed on your PC
- Be licensed to use Aspen Plus

The type library for the server is happ.tlb, located in the Aspen Plus user interface system directory. If you installed Aspen Plus in the default directory, this will be Program Files\AspenTech\AP100UI\xeq. The server is an out-of-process server, apwn.exe.

Before you can access the Aspen Plus type library from Visual Basic, in the Visual Basic Project References dialog box, you must check the Aspen Plus GUI 10.0 Type Library box.

Before you can access the Aspen Plus type library from Excel VBA, in the Excel Tools>References dialog box, you must check the Aspen Plus GUI 10.0 Type Library box.

If Aspen Plus GUI 10.0 Type Library does not exist in the list, click Browse and find the Aspen Plus User Interface system directory. Select happ.tlb.

Error Handling

Errors may occur in calling methods or accessing properties of the Aspen Plus objects. It is important to create an error handler for all code which accesses an automation interface. An automation interface may return a dispatch error for many reasons, most of which do not indicate fatal or even serious errors.

Unless there is an error handler in place any error will normally cause a dialog box to be displayed on the user's screen. In VB the error handler is in the form of an On Error statement, e.g. On Error Goto <line>. It is usual to create an error handling subroutine which will tidy up and exit the application cleanly if any severe errors are encountered.

Viewing the Properties and Methods of Aspen Plus Objects

The properties and methods of the Aspen Plus objects may be viewed in the Automation Client Object Browser:

In Visual Basic 5 and Excel, from the View menu, click Object Browser.

In Excel, the Module sheet must be active for this menu item to be present.

Most of the properties of Aspen Plus objects may be set through the Automation interface to modify the simulation problem. However some properties of simulation objects are read-only. If a property is read-only this is shown in the VB Object Browser, but not in the Excel VBA Object Browser.

Objects Exposed by Aspen Plus

The object exposed by Aspen Plus is the **IHapp** object. An Object Browser may show the class Happ in the place of IHapp. IHapp is the only object type that the class Happ supports. An Aspen Plus application object should be declared as an IHapp object not as a Happ object. Through this object, the other objects and their properties and methods may be accessed.

The objects exposed by Aspen Plus are as follows:

Object	Description
IHapp	The Aspen Plus client object
IHNode	The Aspen Plus problem input and results data are exposed as a tree structure composed of IHNode objects
IHNodeCol	Each IHNode object may own other nodes, and these are organized in an IHNodeCol collection object
IHAPEngine	This object provides an interface to the Aspen Plus simulation engine

The IHapp Object

The IHapp object is the principal object exposed by Aspen Plus. This object provides methods and properties such as:

- Opening a simulation problem
- Controlling the visibility of the Aspen Plus GUI
- Saving a problem

Example of Opening A Simulation

The following VB example obtains the simulation object for an existing simulation problem stored in the backup file pfdtut.bkp, and sets the **Visible** property to display the Aspen Plus graphical user interface.

```
Function OpenSimulation() As IHApp
Dim ihAPSim As IHApp
On Error GoTo ErrorHandler
' open existing simulation
Set ihAPSim = GetObject("C:\Program
Files\AspenTech\Apui100\xmp\pfdtut.bkp")
' display the GUI
ihAPSim.Visible = True
Set OpenSimulation = ihAPSim
Exit Function
ErrorHandler:
MsgBox "OpenSimulation raised error " & Err & ": " & Error(Err)
End
End Function
```

The effect of the GetObject reference is to create a process running the Apwn.exe object server. Any references to the same problem file from the same or other processes will connect to the same running instance of the Apwn server.

The Aspen Plus Tree Structure

The input and results data in an Aspen Plus simulation problem are organized in a tree structure.

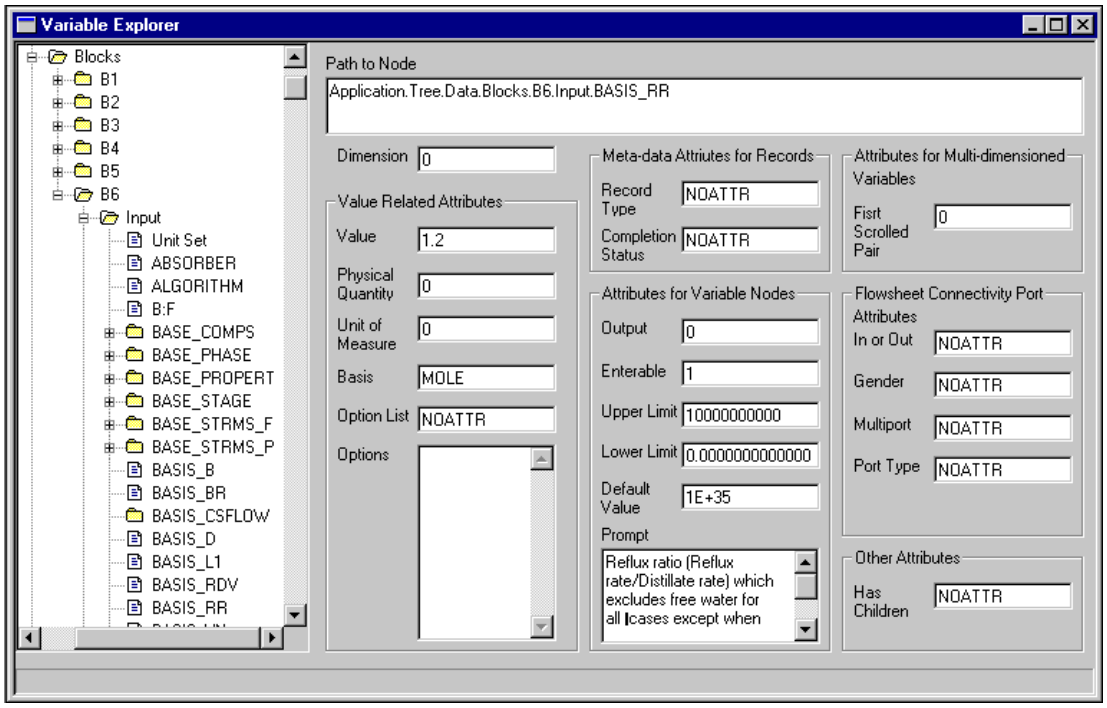
In order to access the data of interest in an Aspen Plus simulation, you need to understand and navigate through the tree structure and locate and identify the variables of interest. To do this, you can use the Variable Explorer in the Aspen Plus User Interface.

Using the Variable Explorer to Navigate the Tree Structure

Use the Variable Explorer to view and access variables associated with your simulation. The Variable Explorer displays the attributes of each variable in the simulation in a similar way to the Data Browser.

To open the Variable Explorer:

↘ From the Tools menu, click Variable Explorer.



The Variable Explorer displays a tree view similar to the Data Browser. The difference is that the Data Browser displays the variables conveniently grouped and laid out on forms with prompt text, scrolling controls, selection boxes and fields for data entry. The Variable Explorer exposes the underlying variables within the simulation problem.

The Variable Explorer is important to the Automation user because it shows the names and the structure of the variables which may be accessed through the Automation interface.

Note The Variable Explorer is read-only. You cannot use the Variable Explorer to change values or other attributes of variables.

Note If you navigate through the tree structure in the Variable Explorer, it is possible to create new objects which you may not be able to delete. For this reason, you should save your Aspen Plus run before using the Variable Explorer and **not** save it after you use the Variable Explorer.

Example of Using the Variable Explorer

This example gives instructions for using the Variable Explorer to access data in the RadFrac block (Block B6) in pfdtut.bkp.

1. From the Tools menu, click Variable Explorer to open the Variable Explorer.

   The tree view on the left displays just the node labeled Root.

2. Double-click on the Root folder icon or click on the + icon to display the nodes immediately below this: Data, Unit Table and Settings.
3. Expand Data to display the next level of nodes: Setup through to Results Summary.
4. Expand the Blocks icon to reveal a list of blocks on the flowsheet: B1 through B6.
5. Expand B6 to display nodes labeled Input through to Work Results.
6. Expand Input to display a list of nodes labeled Unit Set through to Y_EST.

   These nodes represent the simulation input data for the RadFrac block. For example, below the Input node, the node labeled NSTAGE holds the input value for the number of stages in the column.

7. Click on the Output node to display a list of nodes labeled Unit Set through Y_MS.

   These nodes represent the output data for the RadFrac block. For example, below the Output node, the node labeled BU_RATIO holds the result value for the boilup ratio.

   The Path to Node field of the Variable Explorer displays the path to the node which is currently open. From this field, you can copy and paste directly into your program. To do this, complete these steps:

8. Select the text in the Path to Node field, then click the right mouse button.
9. From the menu that appears, click Copy.
10. Go to your application (for example, Visual Basic or the Excel Module sheet).
11. From the Edit menu, click Paste.

Navigating the Tree Structure in the Automation Interface

The tree structure observed in the Data Browser is reflected in the Automation interface.

The objects in an Aspen Plus simulation are exposed as a tree structure of IHNode node objects. The root node of the tree is obtained by the Tree property of IHapp.

Each IHNode object may have zero or more offspring IHNode objects. Each IHNode object has a Dimension property which determines how the offspring nodes are organized. A leaf node (i.e. one with no offspring) has a Dimension of zero.

The offspring nodes of a node object may be obtained as a collection object, IHNodeCol from the Elements property of an IHNode object.

To illustrate this, consider the following example:

```
Sub GetCollectionExample(ihAPsim As IHAPP)
' This example illustrates use of a collection object
Dim ihRoot As ihNode
Dim ihcolOffspring As IHNodeCol
Dim ihOffspring As ihNode
Dim strOut As String
On Error GoTo ErrorHandler
'get the root of the tree
Set ihRoot = ihAPsim.Tree
'now get the collection of nodes immediately below the Root
Set ihcolOffspring = ihRoot.Elements
For Each ihOffspring In ihcolOffspring
   strOut = strOut & Chr(13) & ihOffspring.Name
Next
MsgBox "Offspring nodes are: " & strOut, , "GetCollectionExample"
Exit Sub
ErrorHandler:
MsgBox "GetCollectionExample raised error" & Err & ": " & Error(Err)
End Sub
```

The collection object ihcolOffspring contains the collection of nodes immediately below the root, i.e. those nodes with the labels Data, Unit Table and Settings as observed in the Variable Explorer.

Nodes within each collection object may be accessed in one of two ways:

- You can iterate through the collection object using a **For Each ... Next** structure, accessing each node in turn.
- You can access a node explicitly using the **Item** property of the IHNodeCol object. In order to identify a particular item in a collection, the Item property takes one or more arguments. Each argument is either a string specifying the **label** or **item name** of an offspring node in the next level of the tree, or an integer specifying the ordinal number of the node in the collection of offspring nodes. The number of arguments required to the Item property is given by the Dimension property of the parent.

Thus:

```
For Each ihOffspring In ihcolOffspring
.
.
.
Next
```

iterates through each node in the ihcolOffspring collection, and

```
Set ihDataNode = ihcolOffspring.Item("Data")
```

obtains the node with the label “Data”. Note that the item names are case sensitive.

The Dimension property determines the number of arguments required:

```
if ihcolOffspring.Dimension = 1 then
   Set ihDataNode = ihcolOffspring.Item("Data")
else if ihcolOffspring.Dimension = 2 then
   Set ihDataNode = ihcolOffspring.Item("Data","id2")
endif
```

The Item property is the default property of IHNodeCol, so this statement may be abbreviated simply by writing:

```
Set ihDataNode = ihcolOffspring(“Data”)
```

Dot Notation for Navigating the Tree

To navigate down the tree you can chain the Item property references together. For example, to get to the node labeled NSTAGE which represents the number of stages in a RadFrac block:

```
Set ihNStageNode =      ihAPsim.Elements("Data"). _
                  Elements("Blocks").Elements("B6"). _
                  Elements("Input").Elements("NSTAGE")
```

A more concise notation is also available to navigate down the tree. This simply allows the item names to be chained together, without specifying either the Elements or the Item properties. For example, the above assignment may be written:

```
Set ihNStageNode  = ihAPSim.Tree.Data.Blocks.B6.Input.NSTAGE
```

However, although this 'dot' notation is convenient in many situations it has some restrictions:

- It will only work if the item names are consistent with the syntax of an identifier within the language used by the automation client, in this example Visual Basic. So the item name must not contain embedded spaces or special characters. For example the item name "Unit Table" would be invalid in this notation.
- Certain node types do not support the dot notation. The node types that do not support dot notation are connection, port, setting table, route, label, & unit table.

Data Values

Once you have the leaf node containing the data value of interest, you can obtain the data value associated with the node from the **Value** property. Data values have an associated data type which is held in the **ValueType** property.

ValueType returns one of the following:

ValueType	Description	Visual Basic Data Type
0	Value not defined	
1	Integer	Long
2	Real	Double
3	String	String
4	Node	IHNode

Aspen Plus returns 32bit integer and 64bit real values. Therefore when using Visual Basic, integer and real valued properties should be assigned to Long and Double variables respectively in order to avoid potential overflow errors.

Example of Accessing Data Values

Navigate to and display the number of stages in a RadFrac column (an input data value) and the boilup ratio (a results data value) in a message box.

```
Sub GetScalarValuesExample(ihAPsim As IHAPP)
' This example retrieves scalar variables from a block
Dim ihColumn As ihNode
Dim nStages As Long
Dim buratio As Double
On Error GoTo ErrorHandler
' navigate the tree to the RADFRAC block
Set ihColumn = ihAPsim.Tree.Data.Blocks.B6
' Get the number of stages
nStages = ihColumn.Input.Elements("NSTAGE").Value
' get the boilup ratio
buratio = ihColumn.Output.Elements("BU_RATIO").Value
MsgBox "Number of Stages is: " & nStages _
     & Chr(13) & "Boilup Ratio is: " & buratio, ,
"GetScalarValuesExample"
Exit Sub
ErrorHandler:
MsgBox "GetScalarValuesExample raised error" & Err & ": " &
Error(Err)
End Sub
```

Node Attributes

You can obtain information called **attributes** about the node from the **AttributeValue** and **AttributeType** properties. These take an **attribute number** argument which is an enumerated value from the **HAPAttributeNumber** class.

These tables show some commonly used attributes and their descriptions.

The Attribute Name corresponds to the field in the Variable Explorer.

You can see the full range of possible values and descriptions in the Object Browser of your Automation client (e.g., VB5). In general, you will only need a small subset of the attributes.

Each node typically only supports a subset of the attributes. You can check whether an attribute is supported by querying the AttributeType for the attribute. The attribute types returned are as shown above for ValueType. If the AttributeType property returns a value of zero for an attribute then the attribute is not defined for that node.

Value-related Attributes

Attribute Name	HAP_AttributeNumber	Description
Value	HAP_VALUE	The current value
Physical Quantity	HAP_UNITROW	The row in the Unit Table for the physical quantity of the value
Units of Measure	HAP_UNITCOL	The column in the Unit Table for the physical quantity of the value
Basis	HAP_BASIS	The basis e.g. MOLE or MASS for a value
Option List	HAP_OPTIONLIST	A node whose offspring contain the valid values for this node

Meta-data Attributes for Records

Attribute Name	HAP_AttributeNumber	Description
Record Type	HAP_RECORDTYPE	If the node is record structured, e.g. a block or a stream, this property is a string containing the record type, e.g RADFRAC for a RADFRAC block and MATERIAL for a material stream.
Completion Status	HAP_COMPSTATUS	Returns an integer code giving completion status. Bit masks for interpretation are available in the enum HAPCompStatusCode.

Attributes for Variable Nodes

Attribute Name	HAP_AttributeNumber	Description
Output	HAP_OUTVAR	Is the variable node a results variable (read-only)
Enterable	HAP_ENTERABLE	Can the value attribute be modified?
Upper Limit	HAP_UPPERLIMIT	The upper limit on the value attribute.
Lower Limit	HAP_LOWERLIMIT	The lower limit on the value attribute.
Default Value	HAP_VALUEDEFAULT	The default value for the value attribute.
Prompt	HAP_PROMPT	A descriptive prompt for the node.

Attributes for Multi-dimensioned Variables Nodes

Attribute Name	HAP_AttributeNumber	Description
First Scrolled Pair	HAP_FIRSTPAIR	If the variable uses paired scrolling, the 1 based index of the first item of the pair.

Flowsheet Connectivity Port Attributes

Attribute Name	HAP_AttributeNumber	Description
In or Out	HAP_INOUT	Is the port node an inlet or outlet? For blocks 0 = Inlet 1 = Outlet For Streams: 0 = Outlet 1 = Inlet
Gender	HAP_PORTSEX	Block or Stream port type: 0 = Stream 1 = Block
Multiport	HAP_MULTIPORT	Can the port node be connected to multiple streams? 0=No 1=Yes
Port Type	HAP_PORTTYPE	The type of the port node. 1 = Material 2 = Heat 3 = Work

Other Attributes

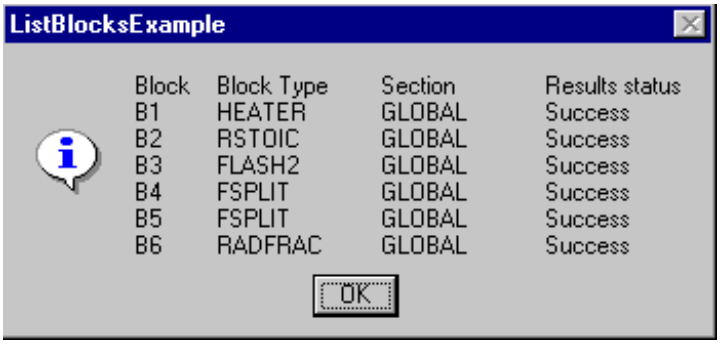
Attribute Name	HAP_AttributeNumber	Description
Has Children	HAP_HASCHILDREN	Returns True if the node has offspring nodes.

Example of Using AttributeValue

The following example subroutine uses AttributeValue to display a list of blocks showing the block type, flowsheet section, and status for each block.

```
Sub ListBlocksExample(ihAPSim As IHApp)
' This example ilustrates retrieving a list of blocks and their
attributes
Dim ihBlockList As IHNodeCol
Dim ihBlock As ihNode
Dim strOut As String
On Error GoTo ErrorHandler
Set ihBlockList = ihAPSim.Tree.Data.Blocks.Elements
strOut = "Block" & Chr(9) & "Block Type" _
         & Chr(9) & "Section  " & Chr(9) & "Results status"
For Each ihBlock In ihBlockList
strOut = strOut & Chr(13) & ihBlock.Name & Chr(9) & _
    ihBlock.AttributeValue(HAP_RECORDTYPE) & "  " & Chr(9) & _
    ihBlock.AttributeValue(HAP_SECTION) & Chr(9) & _
    Status(ihBlock.AttributeValue(HAP_COMPSTATUS))
Next ihBlock
MsgBox strOut, , "ListBlocksExample"
Exit Sub
ErrorHandler:
MsgBox "ListBlocksExample raised error" & Err & ": " & Error(Err)
End Sub

Function Status(CompStat As Integer) As String
' This function interprets a status variable and returns a string

If ((CompStat And HAP_RESULTS_SUCCESS) = HAP_RESULTS_SUCCESS) Then
   Status = "Success"
ElseIf ((CompStat And HAP_RESULTS_ERRORS) = HAP_RESULTS_ERRORS) Then
   Status = "Errors"
ElseIf ((CompStat And HAP_RESULTS_WARNINGS) = HAP_RESULTS_WARNINGS)
Then
   Status = "Warnings"
ElseIf ((CompStat And HAP_NORESULTS) = HAP_NORESULTS) Then
   Status = "No results"
ElseIf ((CompStat And HAP_RESULTS_INCOMPAT) = HAP_RESULTS_INCOMPAT)
Then
   Status = "Incompatible with input"
ElseIf ((CompStat And HAP_RESULTS_INACCESS) = HAP_RESULTS_INACCESS)
Then
   Status = "In access"
End If
End Function
```

This example displays the following message box.



Physical Quantities and Units of Measure

For a value which represents a physical quantity, there are two important attributes:

- The physical quantity (for example, temperature or pressure)
- The units of measurement in which the physical quantity is expressed (for example, degrees Kelvin or degrees Fahrenheit)

The following sections describe how to:

- Retrieve the physical quantity and the units for a value
- Convert a value to a different units of measurement
- Change the units in the Aspen Plus run

Retrieving Units of Measure

You can retrieve:

- Units of measure for a value as a string
- Physical quantity and units of measure as references to the Unit Table

Units of Measure as a String

The unit of measurement symbol for a value can be obtained from the **UnitString** property.

Example of Using Units of Measure

The following subroutine uses the UnitString property to display the outlet pressure of a flash block together with the unit of measurement.

```
Sub UnitStringExample(ihAPSim As IHApp)
' This example retrieves the units of measurement symbol for a
' variable
Dim ihPresNode As ihNode
On Error GoTo ErrorHandler
Set ihPresNode = ihAPSim.Tree.Data.Blocks.B3.Output.B_PRES
MsgBox "Flash pressure is: " & ihPresNode.Value & Chr(9) & _
        ihPresNode.UnitString, , "UnitStringExample"
Exit Sub
ErrorHandler:
MsgBox "UnitStringExample raised error " & Err & ": " & Error(Err)
End Sub
```

The Units Table

Physical quantities and the corresponding units of measurement are described in Aspen Plus by references to a **Unit Table**. Sometimes it is convenient to use the units table directly, instead of dealing with the UnitString of a particular value.

The unit table consists of rows representing physical quantities and columns representing the units of measurement in which the quantities can be expressed. The unit table is exposed in the automation interface below the root node as a node labeled “Unit Table”. The elements in the collection below the Unit Table node represent the rows of the table i.e. physical quantities. The labels of these nodes are the names of the physical quantities. Below each physical quantity node is a collection of nodes whose labels are strings representing the symbols of the units of measurement in which the owning physical quantity may be expressed.

For a node in the tree containing a physical value, the physical quantity, or Unit Table row number, is obtained by reference to the property AttributeValue(HAP_UNITROW). The unit of measurement symbol, or Unit Table column number, is referenced by the property AttributeValue(HAP_UNITCOL). Note that the attribute values are actual row and column numbers and that when referencing the row numbers with the Unit Table collections, you must subtract one from these values.

Converting the Units of Measure for a Value

You can retrieve a value in a specific unit with the **ValueForUnit** property. The **ValueForUnit** property takes two arguments, the desired unit row and the desired unit column.

Example of Converting Units of Measure

Retrieve the pressure of block B3, both in the units specified in the run (psi) and in atm. atm is column 3 in the Unit Table.

```
Sub UnitsConversionExample(ihAPSim As IHApp)
' This example retrieves a value both in the display units and an alternative
Dim ihPres As ihNode
Dim nRow As Long
Dim nCol As Long
Dim strDisplayUnits As String
Dim strConvertedUnits As String
On Error GoTo ErrorHandler
Set ihPres = ihAPSim.Tree.Data.Blocks.B3.Output.B_PRES
' retrieve the attributes for the display units (psi)
nRow = ihPres.AttributeValue(HAP_UNITROW)
nCol = ihPres.AttributeValue(HAP_UNITCOL)
strDisplayUnits = UnitsString(ihAPSim, nRow, nCol)
'select the alternative unit table column (atm)
nCol = 3
strConvertedUnits = UnitsString(ihAPSim, nRow, nCol)
MsgBox "Pressure in Display units: " & ihPres.Value & _
       " " & strDisplayUnits & Chr$(13) & _
       "Pressure in Converted units: " & _
       ihPres.ValueForUnit(nRow, nCol) & " " & strConvertedUnits, _
       , "UnitsConversionExample"
Exit Sub
ErrorHandler:
MsgBox "UnitsConversionExample raised error " & Err & ": " & Error(Err)
End Sub
Public Function UnitsString(ihAPSim As IHApp, nRow As Long, nCol As Long)
' This function returns the units of measurement symbol given
' the unit table row and column
On Error GoTo UnitsStringFailed
UnitsString = ihAPSim.Tree.Elements("Unit Table"). _
              Elements(nRow - 1).Elements.Label(0, nCol - 1)
Exit Function
UnitsStringFailed:
UnitsString = ""
End Function
```

Changing the Units of Measure for the Aspen Plus Run

You can use the HAP_UNITCOL attribute to directly change the units of measurement in the Aspen Plus run.

Changing the HAP_UNITCOL attribute value has a different effect depending on whether the value in an Input or Output value, as follows:

- Changing the HAP_UNITCOL attribute of an output value will convert the retrieved output value into the selected unit of measurement. This is equivalent to changing the units on a Results sheet in the Aspen Plus GUI.
- Changing the HAP_UNITCOL attribute for an input value node will change the input specification units. It does not convert the value into the selected unit of measurement. This is equivalent to changing the units on an Input sheet in the Aspen Plus GUI.

Example of Changing Units of Measure

```
Sub UnitsChangeExample(ihAPsim)
Dim ihPres As ihNode
On Error GoTo ErrorHandler
Set ihPres = ihAPsim.Tree.Data.Blocks.B3.Output.B_PRES
MsgBox "Pressure in default units: " _
             & ihPres.Value _
             & Chr(9) & ihPres.UnitString
' change units of measure to bar
ihPres.AttributeValue(HAP_UNITCOL, True) = 5
MsgBox "Pressure in selected units: " _
             & ihPres.Value _
             & Chr(9) & ihPres.UnitString
Exit Sub
ErrorHandler:
MsgBox "UnitsChangeExample raised error " & Err & ": " & Error(Err)
End Sub
```

Referencing Non-Scalar Variables in the Automation Interface

Most of the data in a simulation problem is organized into arrays, lists or tables, and therefore is contained in multi-valued variables.

Non-scalar data is accessed through the automation interface one value at a time via the Value property of a leaf node. The organization of the nodes which yield the values depends both upon on the number of identifiers required to identify the value, and upon the context. For example:

- A value in a column temperature profile requires the variable name and one additional identifier: the stage number.
- A value in a column composition profile requires the variable name and two additional identifiers: the stage number and the component.
- A reaction coefficient within a reactor requires the variable name and three additional identifiers: the reaction number, the component and the substream

Once a multi-valued variable node is located, selection of identifiers to reach the required individual value involves traversing down the tree. In some cases a single node traversal represents selection of a single identifier. In other cases traversal of a node represents selection of more than one identifier. Each node has the property **Dimension**. If the value of Dimension is > 0 then the node has offspring. The value of the Dimension property for a node determines the number of identifiers associated with an offspring of that node. Dimensions are referenced by an offset; the first dimension number is zero and the last dimension number is the value of the Dimension property minus one.

Offspring nodes are obtained in one of two ways:

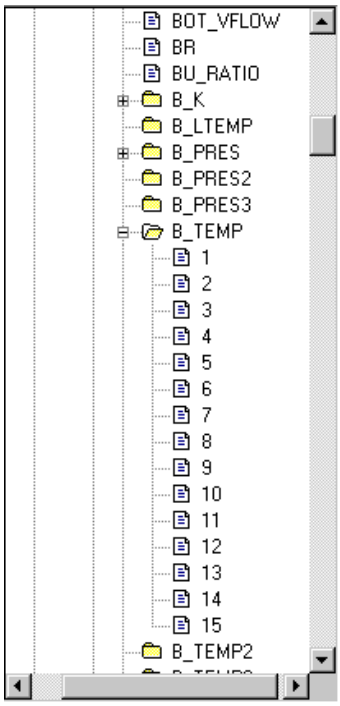
- Using an iterator to loop through the collection object. For example, using a For Each loop in Visual Basic.
- Using the **Item** property of the collection object and specifying an argument for each Dimension of the collection. The argument may be either:
  - An integer **Location** (also known as **RowNumber**) which represents the ordinal number within the dimension. The first Location in each dimension is numbered zero.
  - A string **Label** which identifies the offspring node within in the Dimension.

For each dimension you can obtain the number of valid locations or labels from the **RowCount** property of the collection.

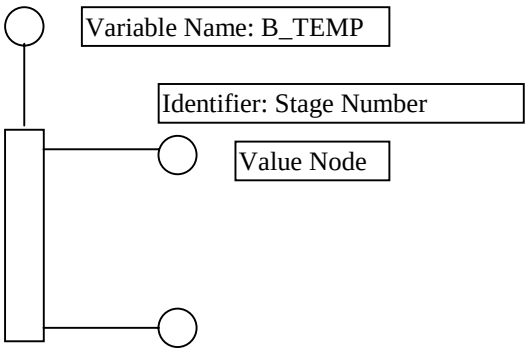
Accessing Variables With a Single Identifier: Column Temperature Profile

The temperature profile in a RadFrac column is an example of a variable with a single identifier. For the pfdtut simulation results, the temperature profile is displayed in the Data Browser in tabular form under Blocks>B6>Profiles>TPFQ.

The same information is located in the Variable Explorer under the Root>Data>Blocks>B6>Output>B_TEMP variable node. Under this node there are fifteen leaf nodes labeled 1 through 15, corresponding to the temperatures on the stages.



The structure of the information is shown schematically below:



Example Showing How to Access Column Temperature Profile through the Automation Interface

To obtain the B_TEMP variable node:

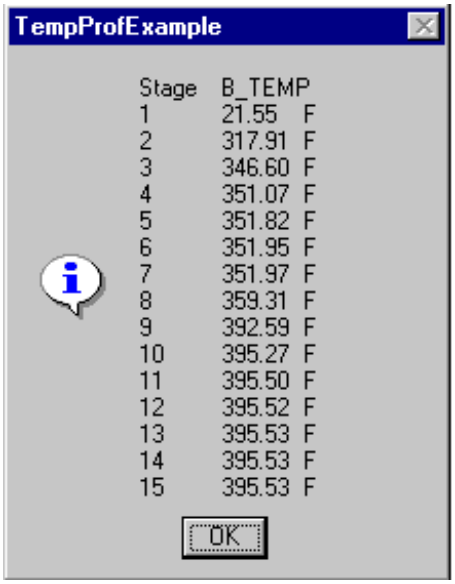
```
Set ihTVar =ihAPSim.Tree.Data.Blocks.B6.Output.B_TEMP
```

Next, create a simple iteration loop to access the offspring nodes representing the stages.

```
For Each ihStage In ihTVar.Elements
.
.
Next ihStage
```

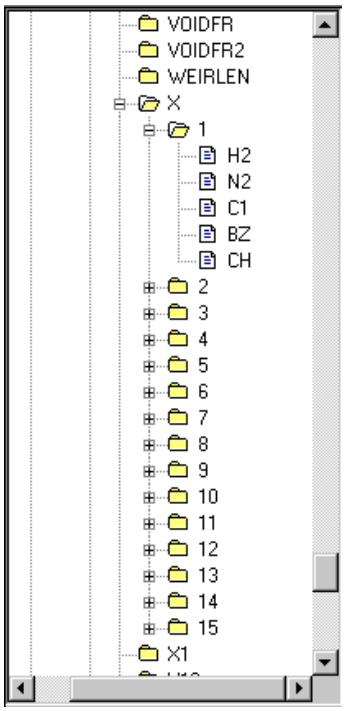
The identifier representing the stage number is retrieved by the Name property of the stage node. The temperature value is retrieved from the Value property of the stage node.

```
Sub TempProfExample(ihAPsim)
' This example retrieves values for a non-scalar variable with one identifier
Dim ihTVar As ihNode
Dim ihStage As ihNode
Dim strOut As String
On Error GoTo ErrorHandler
Set ihTVar = ihAPsim.Tree.Data.Blocks.B6.Output.B_TEMP
strOut = ihTVar.Elements.DimensionName(0) & Chr(9) & ihTVar.Name
For Each ihStage In ihTVar.Elements
   strOut = strOut & Chr(13) & ihStage.Name _
   & Chr(9) & Format(ihStage.Value, "###.00") _
   & Chr(9) & ihStage.UnitString
Next ihStage
MsgBox strOut, , "TempProfExample"
Exit Sub
ErrorHandler:
MsgBox "TempProfExample raised error " & Err & ": " & Error(Err)
End Sub
```

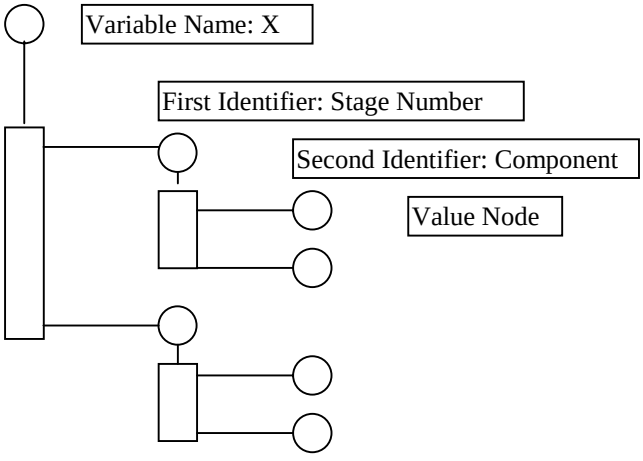


Accessing Variables with 2 Identifiers: Column Composition Profile

The liquid composition profile for RadFrac is an example of a variable with two identifiers. For the pfdtut simulation results, the variable X in the Variable Explorer tree view is shown in this diagram.



The first level of nodes below the variable X represents the stages in the column and each node has the Name property set to the stage number. The second level of nodes contains the nodes for each of the component compositions and the Name property of these nodes is the component id. The Value property of the second level node is the composition of the component in the stage represented by the first level node. This is shown schematically below.



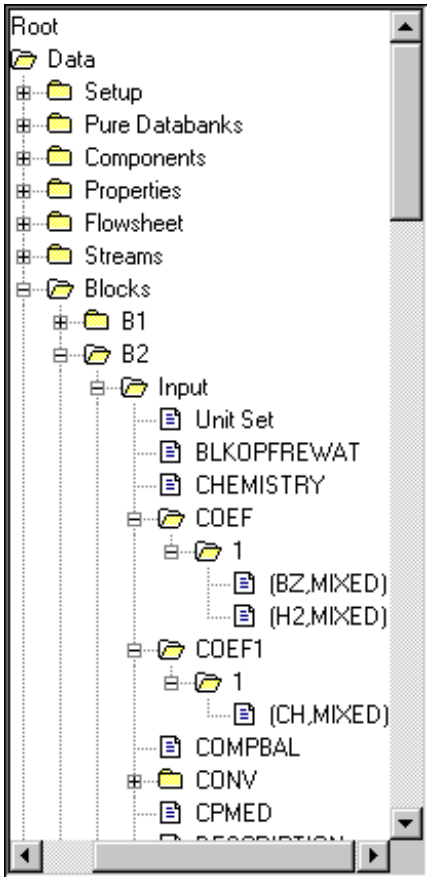
The following code fragment illustrates how to retrieve the component compositions from this structure. It contains two nested loops which iterate through the levels to access the value nodes.

```
 Public Sub CompProfExample(ihAPsim As IHApp)
' This example retrieves values for a non-scalar variable with two
' identifiers
Dim ihTrayNode As ihNode
Dim ihXNode As ihNode
Dim ihCompNode As ihNode
Dim strOut As String
On Error GoTo ErrorHandler
Set ihXNode = ihAPsim.Tree.Data.Blocks.B6.Output.Elements("X")
For Each ihTrayNode In ihXNode.Elements
   For Each ihCompNode In ihTrayNode.Elements
      strOut = strOut & Chr(13) & ihTrayNode.Name & _
               Chr(9) & ihCompNode.Name & Chr(9) & _
               ihCompNode.Value
   Next ihCompNode
Next ihTrayNode
MsgBox strOut, , "CompProfExample"
Exit Sub
ErrorHandler:
MsgBox "CompProfExample raised error " & Err & ": " & Error(Err)
End Sub
```

Accessing Variables With 3 Identifiers: Reaction Coefficients

The following diagram shows the Variable Explorer tree view for the the RStoic reactor block B2 in the pfdtut simulation.

In the RStoic reactor model, the stochiometric coefficients of the reactions are held in the input variables COEF and COEF1 which represent the reaction coefficients for the reactants and products, respectively.

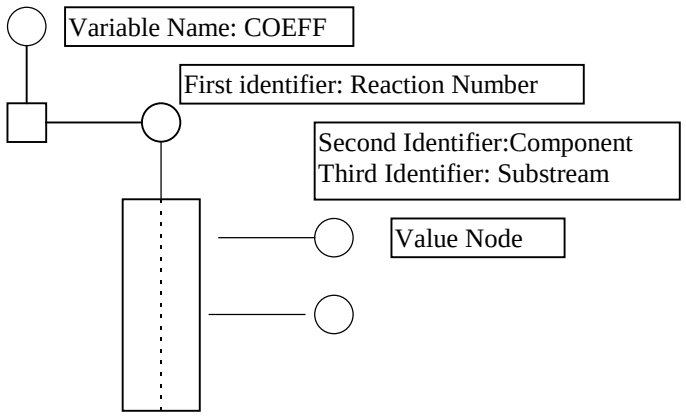


Both these nodes have a list of offspring nodes, each of which represents a reaction equation.

As this block has only one reaction, both COEF and COEF1 have just one offspring node labeled “1” representing the single reaction with the reaction number “1”.

The reaction node has two dimensions so the Dimension property of this node returns a value of 2. There are two identifiers associated with each offspring. The identifier for the first dimension is the component of the reactant. The identifier for the second dimension is the substream, in this case the MIXED substream.

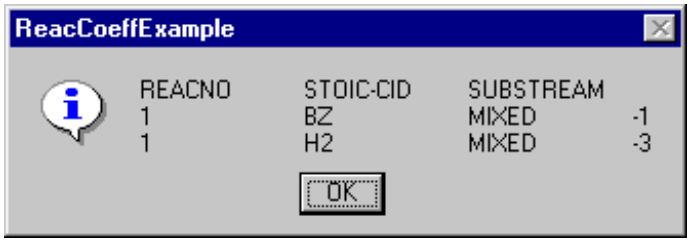
This structure is shown schematically in the following diagram.



The reaction node is an example of a node which uses paired scrolling of identifiers. Here the only significant offspring nodes are those with the same row number in each dimension. The existence of paired scrolling may be determined from the value of the property AttributeValue (HAP_FIRSTPAIR). If the node uses paired scrolling of offspring, this property returns the 1-based index of the first item of the scrolling pair.

The following example code shows how to retrieve the coefficients under the COEF node together with the associated identifiers. Note that because paired scrolling is used, only the nodes with the same value of location in each dimension are accessed.

```
Sub ReacCoeffExample(ihAPsim As IHApp)
' This example illustrates retrieving values for a non-scalar variable
' with three identifiers
Dim ihReacNode As ihNode
Dim ihCoeffNode As ihNode
Dim intOff As Long
Dim strHeading As String
Dim strTable As String
Dim nReacCoeff As Integer
On Error GoTo ErrorHandler
Set ihCoeffNode = ihAPsim.Tree.Data.Blocks.B2.Input.COEF
' loop through reaction nodes
For Each ihReacNode In ihCoeffNode.Elements
  strHeading = ihCoeffNode.Elements.DimensionName(0) _
    & Chr(9) & ihReacNode.Elements.DimensionName(0) _
    & Chr(9) & ihReacNode.Elements.DimensionName(1)
  nReacCoeff = ihReacNode.Elements.RowCount(0)
  ' loop through coefficient nodes retrieving component and substream
  ' identifiers and coefficient values
  For intOff = 0 To nReacCoeff - 1
    strTable = strTable & Chr(13) & ihReacNode.Name & Chr(9) _
    & Chr(9) & ihReacNode.Elements.Label(0, intOff) & Chr(9) _
    & Chr(9) & ihReacNode.Elements.Label(1, intOff) & Chr(9) _
    & Chr(9) & ihReacNode.Elements.Item(intOff, intOff).Value
  Next intOff
  MsgBox strHeading & strTable, , "ReacCoeffExample"
Next ihReacNode
Exit Sub
ErrorHandler:
MsgBox "ReacCoeffExample raised error " & Err & ": " & Error(Err)
End Sub
```



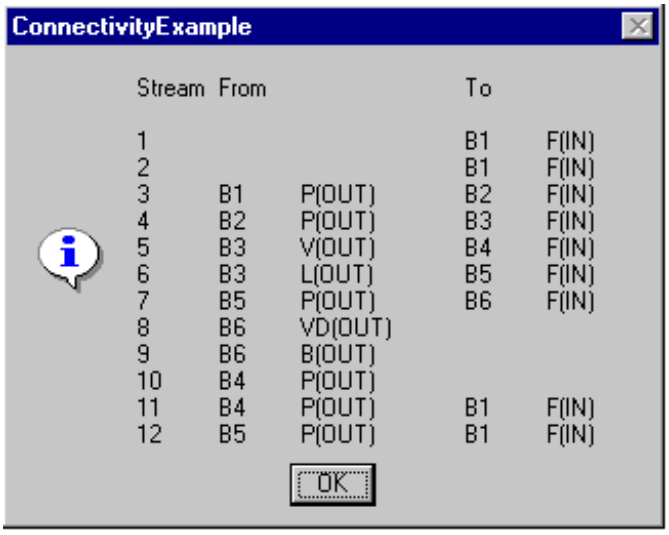
Flowsheet Connectivity and Automation

The connections between blocks and streams in the flowsheet may be accessed via the Automation server.

Both block and stream nodes have Port and Connection nodes which hold information about their ports and what is connected to the ports.

This code sample displays a table showing source and destination blocks and ports for all streams in the flowsheet.

```
Sub ConnectivityExample(ihAPsim As IHApp)
' This example displays a table showing flowsheet connectivity
Dim ihStreamList As ihNode
Dim ihBlockList As ihNode
Dim ihDestBlock As ihNode
Dim ihSourceBlock As ihNode
Dim ihStream As ihNode
Dim strHeading As String
Dim strTable As String
Dim strDestBlock As String
Dim strDestPort As String
Dim strSourceBlock As String
Dim strSourcePort As String
Dim strStreamName As String
Dim strStreamType As String

On Error GoTo ErrorHandler
Set ihStreamList = ihAPsim.Tree.Data.Streams
Set ihBlockList = ihAPsim.Tree.Data.Blocks

strHeading = "Stream" & Chr(9) & "From" _
        & Chr(9) & Chr(9) & Chr(9) & "To" & Chr(13)

For Each ihStream In ihStreamList.Elements
strStreamName = ihStream.Name
strStreamType = ihStream.AttributeValue(HAP_RECORDTYPE)
' get the destination connections
Set ihDestBlock = ihStream.Elements("Ports").Elements("DEST")
If (ihDestBlock.Elements.RowCount(0) > 0) Then
   ' there is a destination port
   strDestBlock = ihDestBlock.Elements(0).Value
   strDestPort = ihBlockList.Elements(strDestBlock). _
   Connections.Elements(strStreamName).Value
Else
   ' it's a flowsheet product
   strDestBlock = ""
   strDestPort = ""
End If
' get the source connections
Set ihSourceBlock = ihStream.Elements("Ports").Elements("SOURCE")
If (ihSourceBlock.Elements.RowCount(0) > 0) Then
   ' there is a source port
   strSourceBlock = ihSourceBlock.Elements(0).Value
   strSourcePort = ihBlockList.Elements(strSourceBlock). _
   Connections.Elements(strStreamName).Value
Else
   ' it's a flowsheet feed
   strSourceBlock = ""
   strSourcePort = ""
End If

strTable = strTable & Chr(13) & strStreamName _
          & Chr(9) & strSourceBlock _
          & Chr(9) & strSourcePort & Chr(9) _
          & Chr(9) & strDestBlock & Chr(9) _
          & strDestPort
Next ihStream
MsgBox strHeading & strTable, , "ConnectivityExample"
Exit Sub

ErrorHandler:
MsgBox "ConnectivityExample raised error" & Err & ": " & Error(Err)

End Sub
```



Controlling a Simulation from an Automation Client

The Engine property of a Happ object returns a IHAPEngine object, which is an interface to the simulation engine. The Happ and IHAPEngine objects provide methods to enable an Automation client program to run and control a simulation.

The following code fragment illustrates how a user is prompted for a simulation parameter, the simulation is re-run and the updated results are displayed to the user.

```
Public Sub RunExample(ihAPsim As IHApp)
' This example changes a simulation parameter and re-runs the simulation
Dim ihEngine As IHAPEngine
Dim nStages As Variant
Dim strPrompt As String
On Error GoTo ErrorHandler
Set ihEngine = ihAPsim.Engine
EditSimulation:
nStages = ihAPsim.Tree.Data.Blocks.B6.Input.Elements("NSTAGE").Value
strPrompt = "Existing number of stages for column B6 = " & nStages _
       & Chr(13) & "Enter new value for number of stages."
nStages = InputBox(strPrompt)
If (nStages = "") Then GoTo finish
' edit the simulation
ihAPsim.Tree.Data.Blocks.B6.Input.Elements("NSTAGE").Value = nStages
' run the simulation
ihAPsim.Run
' look at the status and results
Call ListBlocksExample(ihAPsim)
Call TempProfExample(ihAPsim)
GoTo EditSimulation
finish:
Exit Sub
ErrorHandler:
MsgBox "RunExample failed with error " & Err & Chr(13) & Error(Err)
End Sub
```

Members of Aspen Plus Classes

This section lists the members of each of the exposed Aspen Plus classes.

Members of Class IHApp

Standard VB Properties and Properties to Manipulate the Main Window

Name and Arguments	Member Type	Read-only	Description
Activate()	Sub		Activate the application
Application As Happ	Property	Yes	Returns the application of the object
FullName As String	Property	Yes	Returns the full name of the application
Name As String	Property	Yes	Returns the name of the application
Parent As Happ	Property	Yes	Returns the creator of the object
Visible As Boolean	Property		Returns the visible state of the application

Properties to Access Other Parts of the Object Model

Name and Arguments	Member Type	Read-only	Description
Engine As IHAPEngine	Property	Yes	Return the interface to the simulation engine
Tree As IHNode	Property	Yes	Get top node of file

Basic File Operations

Name and Arguments	Member Type	Read-only	Description
Save()	Sub		Saves current file
SaveAs(filename As String, [overwrite])	Sub		Saves current file under new name
Restore(filename As String)	Property	Yes	Restores, or merges, an archive file into the current problem
WriteArchive(filename As String)	Sub		Exports a archive file.

Initialization Methods to be Used with CreateObject()

Name and Arguments	Member Type	Read-only	Description
InitFromArchive(filename As String)	Sub		Opens an archive and initializes
InitFromFile(filename As String, [readonly])	Sub		Opens a file and initializes
InitFromTemplate (filename As String)	Sub		Opens a template and initializes
InitNew([filename], [overwrite])	Sub		Initializes a new, untitled tree

Basic Run Operations

Name and Arguments	Member Type	Read-only	Description
Reinit()	Sub		Reinitialize the simulation case. To reinitialize specific blocks or streams, use the Reinit member of the IHAPEngine class.
Run()	Sub		Run the simulation case.

Selection Buffer Operations Used to do Cut and Paste Strictly Via Automation

Name and Arguments	Member Type	Read-only	Description
DeleteSelection(Key As String)	Sub		Delete a selection buffer.
NewSelection(Key As String) As IHSelection	Function		Create and return a new selection buffer.
SaveSelection(Key As String)	Sub		Save a selection buffer.
Selection(Key As String) As IHSelection	Property	Yes	Retrieve a selection buffer.

Members of Class IHNode

Standard VB Properties

Name and Arguments	Member Type	Read-only	Description
Application As Happ	Property	Yes	Returns the application of the object.
Parent As Happ	Property	Yes	Returns the creator of the object.

Properties to Access Other Parts of the Object Model

Name and Arguments	Member Type	Read-only	Description
Dimension As Long	Property	Yes	Return the number of dimensions in the directory (0 for scalar).
Elements As IHNodeCol	Property	Yes	Return a collection object containing the node's offspring nodes

Access Data Values

Name and Arguments	Member Type	Read-only	Description
AttributeType (attrnumber As Integer) As Integer	Property	Yes	Get type of attribute for attrnum: 1=int 2=real 3=string 4=node 5=memory block (see Enum HAPAttributeNumber for possible values).
AttributeValue (attrnumbe As Integer, [force])	Property		Get the value of the attribute for attrnum (see Enum HAPAttributeNumber for possible values).
HasAttribute (attrnumber As Integer) As Boolean	Property	Yes	Checks whether attribute is defined for attrnum (see HAPAttributeNumber for possible values).
SetValueAndUnit(Value, unitcol As Integer, [force])	Sub		Store the value attribute and the Unit of Measurement attribute of the object simultaneously.
SetValueUnitAndBasis (Value, unitcol As Integer, basis As String, [force])	Sub		Store the value attribute, the Unit of Measurement attribute, and the basis for the object simultaneously.
Value([force])	Property		Get the value attribute of the object.
ValueForUnit(unitrow As Integer, unitcol As Integer)	Property		Gets the value in the specified units.
ValueType As Integer	Property	Yes	Get type of value attribute: 0=not defined 1=int 2=real 3=string 4=node 5=memory block

Helper Methods

Name and Arguments	Member Type	Read-only	Description
FindNode(path As String) As IHNode	Function		Navigate to a different node.
Name([force]) As String	Property		Returns the name of the object (force argument is unused).
UnitString As String	Property	Yes	Returns the unit of measurement symbol of the node value as a string.

Methods Used to Manipulate the Data

Name and Arguments	Member Type	Read-only	Description
Clear()	Sub		Clear contents of the node.
Delete()	Sub		Delete element.
RemoveAll()	Sub		Remove all elements.

Members of Class IHNodeCol

Standard VB Properties

Name and Arguments	Member Type	Read-only	Description
Application As Happ	Property	Yes	Returns the application of the object.
Parent As Happ	Property	Yes	Returns the creator of the object.

Main Navigation Method

Name and Arguments	Member Type	Read-only	Description
Item(loc_or_name, [loc_or_name2], [loc_or_name3], [loc_or_name4], [loc_or_name5]) As IHNode	Property	Yes	Given a set of indices or names, returns an element in the collection

Principal Data Manipulation Methods

Name and Arguments	Member Type	Read-only	Description
Add([loc_or_name], [loc_or_name2], [loc_or_name3], [loc_or_name4], [loc_or_name5]) As IHNode	Function		Creates and adds a child of type: 1 = scalar 4 = list 5 = named list, with value type of: 0=not defined 1=int 2=real 3=string 4=node 5=memory block.
Insert(element As IHNode, [loc_or_name], [loc_or_name2], [loc_or_name3], [loc_or_name4], [loc_or_name5])	Sub		Inserts an element into collection.

Principal Data Manipulation Methods (Continued)

Name and Arguments	Member Type	Read-only	Description
InsertRow(dimension As Long, location As Long)	Sub		Inserts a new row at location in the specified dimension, dim.
Remove(loc_or_name, [loc_or_name2], [loc_or_name3], [loc_or_name4], [loc_or_name5]) As IHNode	Function		Removes an element.
RemoveRow(dimension As Long, location As Long)	Sub		Removes a row at location in the specified dimension, dim.

Important Properties About the Data

Name and Arguments	Member Type	Read-only	Description
Dimension As Long	Property	Yes	Returns the number of dimensions in the directory.
Label(dimension As Long, location As Long, [force]) As String	Property		Returns the row label for the specified row location in the specified dimension (force argument is unused).
LabelLocation(Label As String, dimension As Long) As Long	Property	Yes	Returns the location, or row number, of the label along the dimension, dim.
RowCount(dimension As Long) As Long	Property	Yes	Returns the number of elements in the dimension.

Other Properties About the Data

Name and Arguments	Member Type	Read-only	Description
Count	Property	Yes	Returns total number of object slots in collection.
DimensionName(dimension As Long) As String	Property	Yes	Gets a display name for the given dimension for variable or table.
LabelNode(dimension As Long, location As Long, [Label]) As IHNode	Property	Yes	Returns a node for manipulating the label.
IsNamedDimension ([dim]) As Boolean	Property	Yes	Returns whether the rows for this dimension of the collection are named.
ItemName(location As Long, [dim], [force]) As String	Property		Returns name or row name for element at location (force argument is unused).

Other Properties About the Data (Continued)

Name and Arguments	Member Type	Read-only	Description
LabelAttribute(dimension As Long, location As Long, attrnum As Integer, [force])	Property		Returns the value of an attribute for the label in the row, location, along the dimension, dim, for attrnum (see HAPAttributeNumber for possible values), (force argument is unused).
LabelAttributeType(dimension As Long, location As Long, attrnum As Integer) As Integer	Property	Yes	Returns the type of an attribute for the label in the row, location, along the dimension, dim, for attrnum (see HAPAttributeNumber for possible values).

Members of Class IHAPEngine

Basic Run Operations

Name and Arguments	Member Type	Read-only	Description
MoveTo(object_type As IAP_MOVETO_TYPE, [object_id])	Sub		Move current simulation step to object or begining of sequence.
Reinit([object_type], [object_id])	Sub		Reinitialize all or portion of simulation (if object_type is used it must be an IAP_REINIT_TYPE.)
Run()	Sub		Run simulation problem.
Stop()	Sub		Stop simulation run.

Manipulate Stop Points

Name and Arguments	Member Type	Read-only	Description
AddStopPoint(type As IAP_STOPPOINT_TYPE, object_id As String, before_or_after As Long)	Sub		Add a stop point, before_or_after: 1 = before 2 = after.
ClearStopPoints()	Sub		Clear all stop points.
DeleteStopPoint(index As Long)	Sub		Delete stop point based on 1-based index.
StopPointCount As Long	Property	Yes	How many stop points are set?
GetStopPoint(index As Long, type As IAP_STOPPOINT_TYPE, object_id As String, before_or_after As Long)	Sub		Retrieve information about a stop point, index: 1-based index of stop point before_or_after: 1 = before , 2 = after.

Manipulate the Client-Server Communications

Name and Arguments	Member Type	Read-only	Description
Host(host_type As Long, [node], [username], [password], [working_directory]) As Boolean	Function		Connect to host specified by host_type (0-based index of available host types).
HostCount As Long	Property	Yes	Returns the number of host types available to connect to.
HostDescription (host_type As Long) As String	Function		Returns a description for the host type specifed by the host_type index (0-based).

Miscellaneous Option Settings

Name and Arguments	Member Type	Read-only	Description
EngineFilesSettings(file As IAP_ENGINEFILES) As String	Property		Retrieve setting for engine files.
OptionSettings(type As IAP_RUN_OPTION) As Boolean	Property		Retrieve setting for simulation run options.

❖ ❖ ❖ ❖

39 Heat Exchanger Design Program Interface

This chapter describes how to use the Aspen Plus heat exchanger design program interface (HTXINT) to transfer heating/cooling curve data from an Aspen Plus run to a heat exchanger design program.

The topics in this chapter include:

- Generating property data
- Starting HTXINT
- Selecting heating/cooling curve results
- Generating the interface file
- Using the interface in a design program

About the Heat Exchanger Design Program Interface

You can use the heat exchanger design program interface (HTXINT) to select heating/cooling curve data from an Aspen Plus run and transfer it to a file in a format that can be read by these heat exchanger design programs:

- B-JAC's HETRAN
- HTFS's TASC, ACOL, and APLE
- HTFS's M-series programs, including M-TASC, M-ACOL, and M-APLE
- HTRI's ST, CST, ACE, PHE, and RKH

You can extend the default data produced by the heating/cooling curves to include all of the properties each design program needs.

Run HTXINT after completing an Aspen Plus run and before starting the design program. HTXINT guides you through a series of prompts. Select the heating/cooling curves for the design program.

HTXINT is an application written using the Aspen Plus summary file toolkit. The location of the FORTRAN source code for your computer is:

Version	Location
OpenVMS	ASP$TOOL
UNIX	$ASPDIR/asptool
DOS	%asptop%\TOOLKIT

Generating Property Data in a Simulation

HTXINT uses property data from heating/cooling curves that can be generated by many Aspen Plus unit operation models. To use HTXINT, you must first use Aspen Plus to generate the required heating/cooling curves. Create one or more heating/cooling curves for each block of interest. For details on specifying heating and cooling curves, see Volume 1, Chapter 14. On the Hcurve form for the block:

1. Specify HXDESIGN in the Property Sets field.
2. Select the required number of points. See Specifying the Number of Heating/Cooling Curve Points, this chapter.
3. Specify the pressure drop.

The following sections describe each step.

Specifying the Property Set

To generate the property data required for all supported heat exchanger program interfaces, select the built-in property set HXDESIGN on the Hcurve forms.

Specifying the Number of Heating/Cooling Curve Points

The Aspen Plus default of ten intermediate points is generally acceptable. You can increase or decrease this number. If the number of points exceeds the maximum number that the heat exchanger program accepts, HTXINT selects the points to include the end-points and any dew or bubble points in the heating/cooling curve. Since Aspen Plus adds extra points for dew or bubble points, more points may be generated than you request.

Specifying Pressure Drops

HETRAN is the only design program that accepts non-isobaric property curves. A heating/cooling curve with a pressure drop cannot be copied to the interface files for the other programs.

HTRI programs accept up to three curves per side at different pressures. For maximum accuracy, define three heating/cooling curves for:

- Inlet pressure
- Outlet pressure
- Intermediate pressure where phase change occurs

Starting HTXINT

To run HTXINT interactively, select the appropriate command:

Version	Command
OpenVMS	HTXINT *runid*
UNIX	htxint *runid*
DOS	HTXINT *runid*

Where:

runid is the ID of the Aspen Plus run

HTXINT reads the data from the summary file named *runid*.SUM (*runid*.sum on UNIX systems). If *runid*.SUM does not exist, HTXINT reads the data from the backup file named *runid*.BKP (*runid*.bkp on UNIX).

The program guides you with a series of prompts. To leave the HTXINT program at any time, type EXIT or QUIT.

HTXINT asks for the name of the target design program:

Please enter the required interface. (BJAC, HTFS, M-HTFS or HTRI) >

Enter the vendor's name for the target design program:

Enter	To write data for
BJAC	HETRAN
HTFS	TASC, ACOL, and APLE
M-HTFS	M-TASC, M-ACOL, and M-APLE
HTRI	ST, CST, ACE, PHE, and RKH

HTXINT prompts for the units of measure:

Please select the units to display the data. (SI, ENG or MET) >

Select the units to use for displaying the heating/cooling curve data on the screen. The options are Aspen Plus units sets.

The B-JAC, HTFS, and HTRI interfaces use the vendor-specific units set closest to the selected Aspen Plus units set, when writing data to the interface file. Different vendors use slightly different combinations of units in their units of measure. The interface uses the most appropriate one. See Using the Interface File in your Heat Exchanger Design Program, this chapter, for a list of the corresponding units of measurement. The M-HTFS interface is always written in SI units.

HTXINT prompts you for the name of the output file:

Please enter the output file name (Default is runid.ext) >>

The default output file is created in your default directory with the same filename as the summary file. The extension depends on the target program. See Using the Interface File in Your Heat Exchanger Design Program, this chapter, for examples of the file extensions.

If you choose the name of a file that already exists, HTXINT asks if you want to overwrite it or choose another filename.

Selecting Heating/Cooling Curve Results to Export

To select the heating/cooling curve:

1. Enter the block ID.
2. Enter the heating/cooling curve number.
3. Choose whether to view or write the data for the heating/cooling curve.

When you complete Step 3, you are returned to Step 1. You can continue to select and copy further heating/cooling curve data, up to the maximum number of heating/cooling curves allowed by the heat exchanger design program.

The following sections describe each step.

Entering the Block ID

HTXINT lists on the screen all blocks that have heating/cooling curves. Enter a block ID to display the list of heating/cooling curves for that block. For example, the screen displays:

```
The following blocks have Hcurves.

+-----------------------------+
| Block ID  |  Model type     |
+-----------+-----------------+
|  E01      |    HEATX        |
|  T01      |    RADFRAC      |
+-----------+-----------------+


Please select a block ID from the list. >
```

Entering the Heating/Cooling Curve Number

Enter the heating/cooling curve number from the list of heating/cooling curves for the block. The screen displays:

```
The Hcurves for block E01 are as follows.

+-------------------------------------------------------------------------+
|NUMBER|Side | Type |   Pressure range    |  Temp. range | Vfrac range|
|      |     |  .   |    (N/SQM)          |      (K)     |            |
+------+-----+------+---------------------+--------------+------------+
|  1   | HOT | HOT  |0.410E+07 0.410E+07 |322.039 207.039|1.000 0.639 |
|  2   |COLD |COLD  |0.445E+07 0.445E+07 |183.150 200.656| 0.000 0.000|
+-------------------------------------------------------------------------+
```

```
Please select an Hcurve from the list by entering its number. > 2
```

Viewing and Writing the Data for the Heating/Cooling Curve

HTXINT asks:

```
Do you wish to view data points for this curve? (Y/N) >
```

If you enter N, the data for this heating/cooling curve is written to the interface file.

If you enter Y to view the data, the heating/cooling curve points appear. For example:

TEMP (K)	DUTY (WATT)	PRES (N/SQM)	VFRAC
322.039	0.31907E+04	0.41024E+07	1.000
317.775	-0.71198E+05	0.41024E+07	1.000
311.584	-0.21327E+06	0.41024E+07	0.994
301.130	-0.46122E+06	0.41024E+07	0.980
290.675	-0.72166E+06	0.41024E+07	0.963
280.221	-0.99223E+06	0.41024E+07	0.941
269.766	-0.12693E+07	0.41024E+07	0.916
259.312	-0.15503E+07	0.41024E+07	0.888
248.857	-0.18344E+07	0.41024E+07	0.857
238.403	-0.21230E+07	0.41024E+07	0.822
227.948	-0.24205E+07	0.41024E+07	0.779
217.493	-0.27370E+07	0.41024E+07	0.724
207.039	-0.30963E+07	0.41024E+07	0.639

Indicate whether you want to write the data for the heating/cooling curve by responding to this prompt:

```
Do you want this Hcurve to be written to the interface file? (Y/N) >
```

If you enter Y, the heating/cooling curve data is written to the output file. If you enter N, you can enter another heating/cooling curve number.

If the heating/cooling curve contains calculation errors, any points with errors are indicated with an asterisk (*) and the following prompt appears:

```
Do you want data for the points with the calculation errors to be
suppressed? (Y/N) >
```

If you enter Y, the points with errors will be excluded.

Generating the Interface File

After you choose a heating/cooling curve to write to the interface file, HTXINT provides prompts for descriptive data for the target design program. Examples of information you may want to provide are a problem description, case name, and hot or cold fluid name. HTXINT indicates when the number of characters you can use is restricted, and validates your data entry.

HTXINT generates the input, then asks you if you want to select another heating/cooling curve.

The following text shows an example of providing descriptive data for the HTRI interface.

```
Please provide a one line description of the problem up to 70
characters.

> HCURVE DATA COPIED FROM HEAT EXCHANGER E01

Please provide a one line case description of up to 70 characters.

> BASE CASE

Enter the cold fluid name. 12 characters maximum > C1

Data for curve has been written to the interface file.

Do you want to select another Hcurve? (Y/N) > N
```

Using the Interface File in Your Heat Exchanger Design Program

The interface program creates a file containing heating/cooling curve data. This file includes the descriptive data you entered at the prompts. The file format, name, and how the file is used depends on the design program. The input file that HTXINT produces contains only heating/cooling curve data. It does not contain the complete data necessary to run the design program.

Design Program	Interface Filename	Units of Measure†	Maximum Number of Heating/Cooling Curves	Maximum Number of Heating/Cooling Curve Points	How to Use
B-JAC	*filename*.bji	SI, US, metric	Hot side = 1 Cold side = 1	13 (inlet, outlet, dew point, bubble point, and up to 9 others)	Run B-JAC. Select Shell&Tube Heat Exchanger Programs, then Thermal Design, then Establish the Filename, then Create or Modify Input File.
HTFS	*filename*.dat	SI, British, metric	Hot side = 1 Cold side = 1	12	Edit and complete the data in the input file.
HTFS M-Series	*filename*.psf††	SI only	Hot side = 1 Cold side = 1	12	Run DOS program. Select Input Data Editor. Then either create or edit input and import the interface file, using function key F5. Or Run Windows program. Select Import Simulator Input File from File menu.
HTRI	*filename*.dat	SI, US, MKH	Hot side = 3 Cold side = 3	10	Edit and complete the data in the input file. Or Enter the program and select the input file.

† *These are the vendors' names for units of measurement sets that correspond approximately to the SI, ENG, and METRIC sets in Aspen Plus.*

†† *Use this file in the way described in this table. The format and units of a process simulator file are different from the usual HTFS input file.*

❖ ❖ ❖ ❖

Index

A

Active links 37-16
  saving files 37-24
ActiveX automation
  using 38-1
Aspen Plus
  automation server 38-1
Attaching images 37-12
Attributes
  node 38-10
AttributeType 38-10
AttributeValue 38-10
Automation
  about 38-2
  accessing column temperature profiles 38-19
  clients 38-27
  controlling simulation 38-27
  error handling 38-2
  example 38-24
  non scalar data 38-17
  tree structure 38-7
  using 38-2
Automation server
  flowsheet connectivity 38-25
  setting up 38-2
  using 38-1, 38-2

B

Blocks
  attaching images 37-12
  creating pressure relief 33-2

C

Calculations
  Pressure Relief 33-27
  stopping 33-25
Copy with Format command 37-3
Copying
  about 37-2
  Copy with Format 37-3
  data 37-2
  plots 37-11
Cutting and pasting
  from applications to Aspen Plus 37-8
  into other applications 37-6
  within Aspen Plus 37-4

D

Data
  copying 37-2
  packages 34-1
  pasting 37-4
Data values
  obtaining 38-9
Deleting
  objects 34-7
Design rules
  Pressure Relief 33-10
Dot notation
  using 38-8
DynamicInput form 33-20
DynamicResults form 33-29

E

Embedded objects are saved as part of a run 37-29
Embedding
  objects 37-27, 37-28
Emergency events
  describing 33-20
Errors
  automation 38-2
Excel
  copying data from 37-8
  updating links 37-26

F

Files
  active links 37-24

with active links 37-25
with embedded objects 37-29
with links 37-25
Fillage
specifying 33-10
Flowsheets
connectivity 38-25
embedding objects 37-27

H

HAP_UNITCOL attribute 38-16
Heat exchanger design program interface
overview 39-2
Hiding
objects 34-7
HTXINT
overview 39-2

I

ID conflicts
resolving 34-3
IHapp object 38-3, 38-4
Input
dynamic 33-20
Inserts
about 34-1
creating 34-2
electrolyte 34-6
importing 34-2
using library 34-6
Interoperability
about 37-1

L

Libraries
inserts 34-6
stream 35-1
Link container
definition 37-24
Link source
definition 37-24
visibility of applications 37-25
Linking
applications 37-16
Linking results 37-17
Links
saving files 37-24
updating in Excel 37-26
Live data links
about 37-2

N

Node attributes
about 38-10
Node objects
tree structure 38-7
Nodes
offspring 38-18
Non scalar data
accessing 38-17

O

Object Browser 38-3
Objects
collection 38-7
embedding 37-27, 37-28
error handling 38-2
hiding 34-7
IHapp 38-3, 38-4
revealing 34-7
tree structure 38-7
viewing properties 38-3
Offspring nodes
obtaining 38-18
OLE automation
about 37-1, 37-2

P

Paired scrolling 38-24
Pasting
about 37-2
Aspen Plus data into applications 37-6
data 37-4
from other applications 37-8
plots 37-12
Pipe model
diameters 33-18
Pipe Schedules tables 33-18
Plots
attaching to blocks 37-12
attaching to flowsheets 37-12
copying to other applications 37-11
pasting onto flowsheet 37-12
Pressure Relief

about 33-2
calculations 33-27
creating block 33-2
devices 33-14
results 33-27
rules 33-10
scenarios 33-3, 33-5
specifying 33-24
stopping calculations 33-25
streams 33-6
venting systems 33-13
Property package
creating 34-5

R

Reactive systems
specifying 33-24
Relief devices
specifying 33-14
Resolve ID Conflicts dialog box 34-3
Results
dynamic 33-29
examining 33-27
linking 37-17
Pressure Relief 33-27
steady state 33-27
viewing 33-29
viewing steady state 33-27
Revealing
objects 34-7
Rules
Pressure Relief 33-10

S

Saving
files with embedded objects 37-29
links 37-24
Scenarios
dynamic 33-8, 33-12, 33-24
pressure relief 33-3
steady state 33-6
Scrolling
paired 38-24
Simuations
controlling from automation client 38-27
Special data packages 34-1
SteadyStateResults form 33-27
Stream libraries
about 35-1
creating 35-2
modifying 35-2
Streams
specifying 33-6
STRLIB
about 35-2
commands 35-3
running in batch mode 35-3
running interactively 35-2

T

Table Format Files
about 36-1
location 36-2
TFF
about 36-1
location 36-2
Tree structure
about 38-4
automation interface 38-7
dot notation 38-8
Variable Explorer 38-5
Troubleshooting
ID conflicts 34-3

U

Unit Table
about 38-15
units of measure
changing 38-16
UnitString property
using 38-14

V

ValueForUnit property 38-15
ValueType property 38-9
Variable Explorer
about 38-5
example of using 38-6
using 38-5, 38-6
Venting systems
specifying 33-13
Vessel Neck
specifying 33-16
Visual Basic
about 37-1