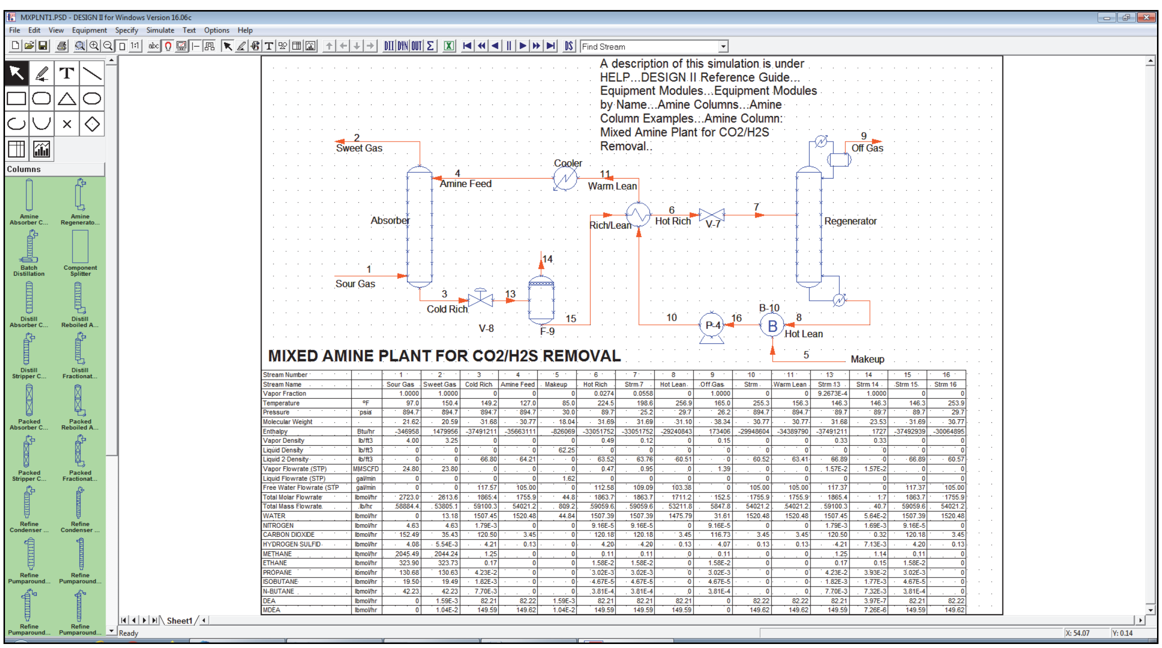
DESIGN II for Windows™

User Guide

Version 16.0



WinSim® Inc.

Advanced Engineering Software

Copyright Notice

Release 16.0, May 2021

WinSim Inc.

8653 FM 2759 Road, Richmond, TX 77469

281-545-9200 Telephone
281-545-9200 x106 Technical Support
281-545-8820 Fax

Web-site: www.winsim.com
E-mail: support@winsim.com
E-mail: sales@winsim.com

Table Of Contents

Chapter 1: Introduction

Welcome to DESIGN II for Windows !

Introducing . . . DESIGN II for Windows

DESIGN II for Windows is the first process simulator exclusively developed to take advantage of the rich graphical environment of Microsoft Windows.

The program is intuitive, allowing you to quickly start drawing flowsheets by pointing and clicking. You can validate your data; then easily run your simulation using a powerful, proven simulator engine: DESIGN II. Stream and equipment results can be instantly transferred to Microsoft Excel spreadsheet (available separately) or to other programs such as word processors for reports.

Who Should Read This Manual?

Welcome to the ***DESIGN II for Windows User Guide***. This manual is written as both a learning tool for beginning users (the basics section) and experienced users (the reference section).

In the basics section, we will introduce you to the DESIGN II for Windows program, guiding you step-by-step on how to use it to get your job done. An on-line tutorial is also available.

After becoming familiar with using DESIGN II for Windows, you can refer to the reference section to find the information you need, or to the on-line help system.

We presume that you are familiar with computers and are at least somewhat familiar with the Microsoft Windows interface. We cover some basics of Windows, but you may want to read the software documentation for your computer system.

Conventions

This manual follows a set of guidelines for the presentation of material. The Font conventions are listed below.

Lightface	Normal text.
Bold, italic	Refers to a specific chapter, section, or manual for more information.
Italic	Used to indicate text you will enter at prompts; file names are also italicized.
Boldface	Used primarily to indicate mouse or pointer actions you will perform; may also be used to emphasize for normal text

How This Manual Is Organized

This manual consists of these chapters:

- Introduction (this chapter)
- Installation (Chapter 2)
- Basics of using DESIGN II for Windows (Chapter 3)
- Options/Preferences (Chapter 4)

- Equipment Modules (Chapters 5 and 6)
- Streams (Chapter 7)
- Annotations and Drawing Elements (Chapter 8)
- General Specifications (Chapter 9)
- Simulation (Chapter 10)
- Results (Chapter 11)

Where Next?

If DESIGN II for Windows **is not** installed on your PC, please turn to the Installation Instructions.

If DESIGN II for Windows **is** installed, please turn to the ***Basics*** section.

DESIGN II Reference Guide

There is also an online manual that details the keyword commands for DESIGN II and ChemTran. Using the extensive search system built into the Windows Help system, you can research the simulator kernel in detail.

Chapter 2: Installation

Prepare to Install

Before installing the DESIGN II for Windows software, make sure you have the necessary hardware and software; then check to be certain you have all the required materials.

Check System Setup

DESIGN II for Windows requires the following hardware / software:

1. Microsoft Windows 10 / 8 / Windows 7 / Vista / XP **(Windows 7 x64 is highly recommended)**
2. A PC with an Intel Pentium processor (any 32 bit PC compatible CPUs with a math coprocessor are also supported) (**a 3.0+ Ghz Intel Quad Core is highly recommended**)
3. 512 MB of RAM memory for Windows XP **(2048 MB is highly recommended**)
4. 4 GB of RAM memory for Windows 7 x86
5. 8 GB of RAM memory for Windows 7 x64 **(16 GB is highly recommended)**
6. 300 MB of free hard disk space for installation, 2,000 MB of free disk space for simulation work area
7. A CD-ROM drive if not installing from the internet or local network
8. USB port for hardware security key (if so licensed)
9. Mouse or other pointing device
10. Windows-compatible graphic monitor
11. If you wish to transfer results data to a spreadsheet; then Microsoft Microsoft Excel 2016, 2013, 2010, 2007, 2003 or later is required.
12. There is not a native version of Design II for Windows for the Macintosh. We have been told by several users that Design II for Windows does run very well under the Parallels software. There are probably other solutions available also.

NOTE: We use Microsoft Excel and Word for Windows as example programs; these are licensed separately through other suppliers.

Check Materials

The following materials have been sent in order to install DESIGN II for Windows:

1. a CD-ROM labeled DESIGN II for Windows (if CD-ROM version)
2. a hardware security device (if using hardware key locking and not software locking)
3. a DESIGN II for Windows User Guide
4. a Network License User Guide (if the Network version is licensed)
5. if you are a licensed user; then the back of your CD-ROM will have a sticker with your serial number. If you have a hardware key; then your sticker will also have your password on it.

Install DESIGN II for Windows

DESIGN II for Windows uses an automated installation procedure designed to work in the Windows environment. The installation procedure will verify your system hardware, create directories as needed, copy the DESIGN II software to the selected location, update your system files and create a "DESIGN II" group with all the correct icons in the Program Manager.

If this is the CD-ROM version then insert the DESIGN II for Windows CD-ROM into your CD-ROM drive. This procedure will use D: to indicate the CD-ROM drive. Continue the installation procedure by clicking the Start button on the Task Bar and selecting Run. Then type D:\SETUP in the Open: field and click the OK button.

The installation procedure will prompt you for the Destination directory before beginning to install the software. The default destination directory is C:\DESIGNII. The drive will be checked for sufficient free space to install the selected options. Verify the directory where DESIGN II for Windows will be installed. The directory will be created if it does not exist.

NOTE: You can terminate the installation procedure at any time before files are installed by clicking the Cancel button on any dialog window.

After you have completed the verifications, the files for DESIGN II for Windows will be installed on your hard drive. A full system installation takes approximately 10 minutes on a Pentium PC. The Program Manager group "DESIGN II" will be created at this time also.

Attach the hardware security device (if licensed and included) to the parallel port LPT1: or a USB port on the computer where DESIGN II for Windows will run. You must shutdown and turn off the computer in order to attach the hardware key.

Software Security - General

In order to prevent mass duplication of DESIGN II for Windows by software pirates (see Software Publishers Association, www.spa.org), we require that DESIGN II for Windows be locked to the PC that it is licensed to.

DESIGN II for Windows has several options for locking the software to a particular PC. The first method is called Software Locking and is used by all test drive and software locking license. The second option for locking the software to a PC is a hardware security key attached to the PC parallel port or a USB port. The third option for locking the software is to a network license server. The fourth option for locking the software is a site license where the password is locked to the serial number, customer name and customer location.

DESIGN II License Dialog

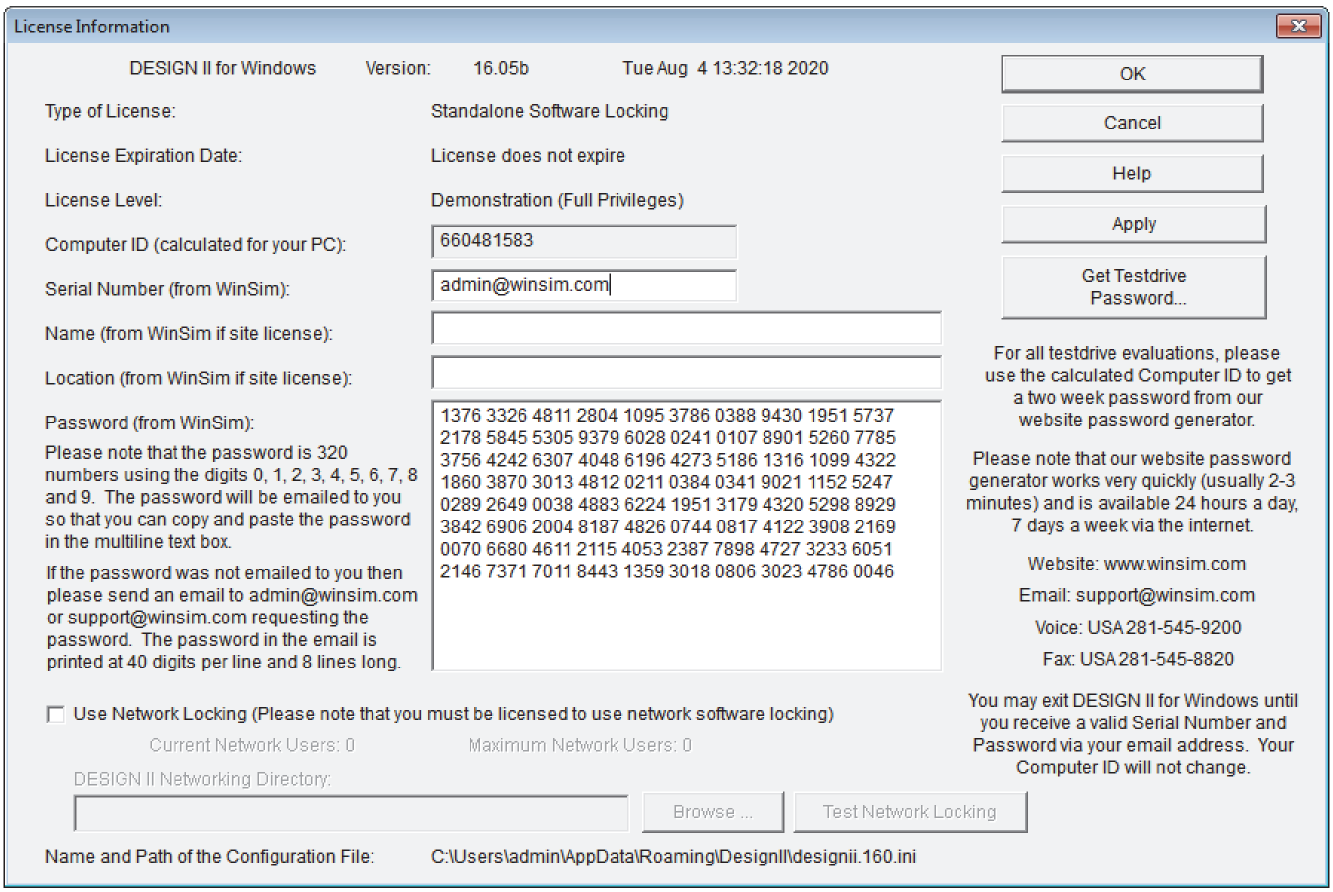


Figure 1: DESIGN II License Dialog

This dialog is for information on the current security of DESIGN II for Windows. You may also modify the software security if your type of license is Software Locking. If you are a licensed user, your serial number and password are on the back of your CD ROM.

Data Item	Description
Type of license:	Type of security used for the software: AH hardware key, BH hardware key, CH hardware key, EH hardware key, Standalone Software Locking, USB hardware key, Site Locking or Network Software Locking
License Expiration Date:	The date that the software will cease to work (if limited by your password)

License Level:	Professional or Basic
Computer ID	A constant number generated by your hard disk serial number to be supplied to Technical Support. The Computer ID is not used for hardware locking for licensed users.
Serial Number	Supplied by Technical Support that enables DESIGN II for Windows to operate on your computer. If you are a licensed user then your serial number will be on the back of your CD-ROM.
Password	Supplied by Technical Support that enables DESIGN II for Windows to operate on your computer. If you are a licensed user then your Password will be on the back of your CD-ROM.
Name	An optional (issued by WinSim for site license) parameter that can be set for your usage.
Location	An optional (issued by WinSim for site license) parameter that can be set for your usage.

Please note that you can get a two week password and serial number from our website, www.winsim.com, at any time.

All parameters entered into this dialog are stored in the c:\designii\designii.ini file for continuous usage.

Software Security - Software Locking

After installation, DESIGN II for Windows will display a dialog requesting that you send a ‘Computer ID’ to our Technical Support for a Password and Serial Number. The Computer ID is a nine digit number generated by certain characteristics of your PC. The Password generated by Technical Support is a 320 digit decimal (0 - 9) number. The Serial Number is an 8 digit hexadecimal number that is permanently assigned to each software license. The standard testdrive serial number is ABC12345.

Please note that you can get a two (2) week Software Locking password from our web site 24 hours a day at http://www.winsim.com/gen2wkps.html.

Software Security - Hardware Locking

The hardware security key is attached to the parallel port of a PC or one of the USB ports (depending on the hardware key type). If your license includes a hardware key then you should have received a Serial Number and Password with your distribution to enter in the DESIGN II for Windows license dialog.

In order to use the Hardware Security Key on Windows Vista, XP or 2000, you must install a special device driver (the port driver). A device driver must be installed in order for DESIGN II for Windows to access the hardware security device. The device driver is not required on Windows Me, 98 or 95.

NOTE: You must have administrator privileges in Windows Vista / XP / 2000 to install the Port Driver.

Please install DESIGN II for Windows on your Windows computer. Then install the Port Driver by:

1. Click the “Start” button. Click on the “Programs” item. Click on the “DESIGN II” group.
2. Click on the “Windows Hardware Key Device Driver”
3. The Windows hardware key device driver will now be automatically installed.

After installing the hardware key on your parallel port or your USB port you should test it by running the KEYIDW32.EXE program in the C:\DESIGNII\PDRIVER directory. This is a command line program to be executed from the command window line.

If you have a parallel port key and the hardware key detection program, keyidw32.exe, does not work then your parallel port may be disabled in your BIOS. Please reboot your PC and enter the BIOS (press the Delete key or the Escape key upon boot). Find the appropriate command screen and ensure that the parallel port is enabled. Many modern PCs such as Toshiba laptops and Intel-based motherboards are showing up from the factory with the parallel port disabled.

In fact, some modern PCs do not even have a parallel port. If this is the case and you own a parallel port key, please contact WinSim Sales for a replacement USB hardware key.

If you continue to have problems then run the Sentinel program “rainbows.exe” in the directory C:\designii\pdriver\.

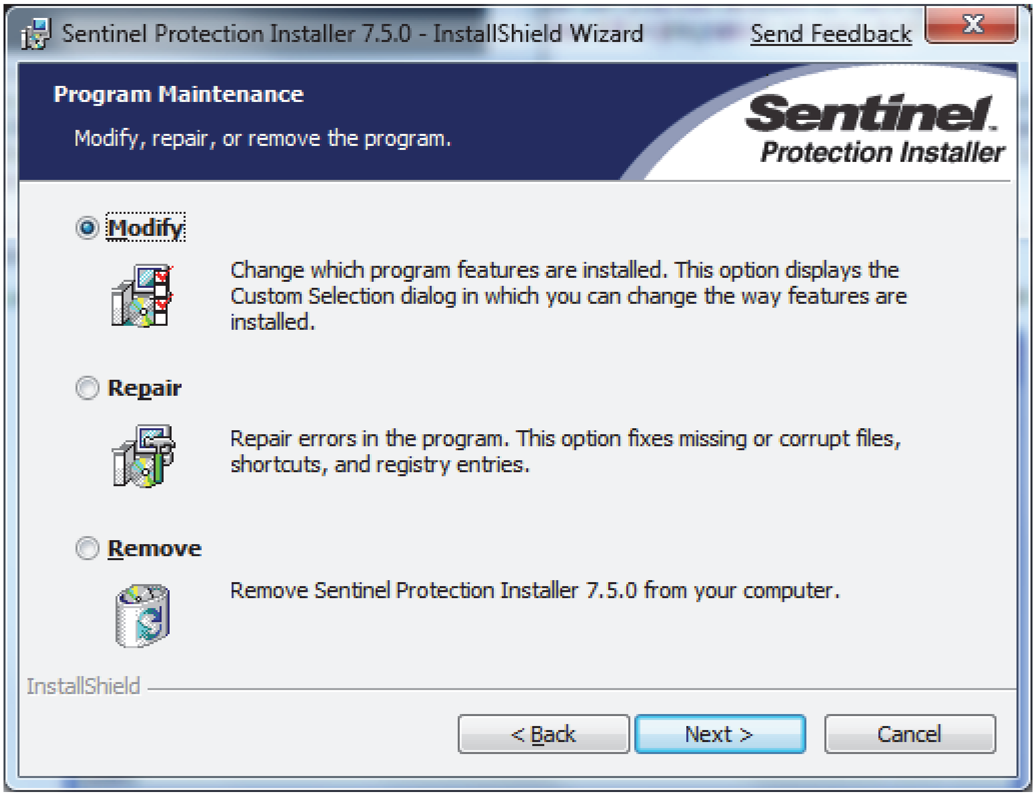


Figure 2: Rainbow Sentinel System Driver

Click on the Repair button. The Sentinel Protection Installer will reinstall itself and ask you to reboot the pc. Please do so.

Software Security - Network Locking

Please see the DIIW Network Locking Guide (you must be licensed for this feature). This guide will be emailed to you from our software administrator (admin@winsim.com).

Software Security - Site Locking

If you are licensed for site locking then you will be issued a password, serial number, name and location. Your password will be locked to your serial number, name and location. All four parameters issued by WinSim must be entered into the License Dialog in order to use DESIGN II for Windows.

Password Expiration

If you are leasing DESIGN II for Windows; then your password will expire on a specific date. You can check this date by opening the License Information Dialog box (see Figure 2 – 1).   The lIcense dialog is located on the File menu.

Please note that DESIGN II has a date rollback check feature that detects any changes in the system clock for expiring licenses.

Other Files on the DESIGN II for Windows CD-ROM

SETUP.EXE	DESIGN II for Windows Installation
README.TXT	Notes on the installation
DEMO\DIIWDEMO.PPT	DESIGN II for Windows Slide Show
DEMO\DIIWHIST.PPT	Slide show on the history of DESIGN II for Windows
DEMO\PPTVW32.EXE	Slide show viewer for Windows 95 \ NT
AUTOPTN*	Files for providing the initial CD menu

Files in the DESIGN II for Windows directory: c:\designii

CHEMTRAN.DLL	ChemTran simulator kernel
CHEMTRAN.EXE	ChemTran command line execution file
CHMWIN32.EXE	ChemTran windowed execution file
CHM.BAT	Command file for running ChemTran
COMPDATA.TXT	Pure Component Database
COMLIST0.DAT	Component list sorted by component ids
COMLIST1.DAT	Component list sorted by name
COMLIST2.DAT	Component list sorted by mixed amine
COMLIST3.DAT	Component list sorted by fomula
COMLIST4.DAT	Component list sorted by molecular weight
COMLIST5.DAT	Component list sorted by boiling point
D2PROPS.EXE	Sample Visual Basic application for calculating physical properties
DEMO.EXE	Demo for DESIGN II for Windows
DESCMDS.DAT	Data file for DESIGNII.DLL
DESIGNII.DLL	DESIGN II simulation kernel
DESIGNII.EXE	DESIGN II command line execution file
DESIGNII.CHM	DESIGN II / ChemTran help file
DESIGNII.INI.TXT	DESIGN II for Windows user configurable parameters file
DESWIN.EXE	DESIGN II for Windows main execution file
DESWIN.CHM	DESIGN II for Windows help file
DII.BAT	Command file for running DESIGN II
DII2VBAS.DLL	Visual basic interface into the DESIGN II simulator kernel
DII2VBAS.XLS	Sample Excel spreadsheet showing how to call DII from VB
DIIWIN32.EXE	DESIGN II windowed execution file
EXPANDER.PSD	Sample flowsheet
FM2NBOOK.EXE	DESIGN II Excel Blaster
FM2NBOOK.INI	DESIGN II Excel Blaster configuration file
GEN_REFERENCE.PDF	DESIGN II / ChemTran General Reference Guide (Adobe Acrobat)
UNIT_MODULE.PDF	DESIGN II Unit Module Reference Guide (Adobe Acrobat)
USER_GUIDE.PDF	DESIGN II for Windows User Guide (Adobe Acrobat)
IDE21201.VXD	Windows 95/98/ME Hard Drive utility
INPUTMAN.EXE	DESIGN II / ChemTran input file editor
METRIC.SYM	Symbols for Metric and US dimensional unit modes
MXPLNT1.PSD	Sample flowsheet
REFINERY.PSD	Sample flowsheet
RELNOTES.CHM	Release Notes
SAMPLE.STY	Sample flowsheet legend layout
SX32W.DLL	Dynamic Link Library for DESIGN II
SURVEY.DOC	User Survey
TEMPLATE.DAT	Keyword commands for various unit modules / options
TEMPLATE.ILF	Inline Fortran command various unit modules
TEMPLATE.XLS	Template file for Excel 97 – 2008 results transfer
TEMPLATE_EXCEL95.XLS	Template file for Excel 95 results transfer
UNWISE.EXE	Removal tool for DESIGN II for Windows
ZIP32.DLL	Dynamic Link Library for DESIGN II and DESIGN II for Windows
BIN\	Visual Basic / Visual C++ / D2Props driver files
PDRIVER\RAINBOWS.EXE	Hardware Key device driver
PDRIVER\KEYIDW32.EXE	Hardware Key test utility
SAMPLES\	Samples Flowsheet and ChemTran input files (See Appendix A)
TUTORIAL\	Files for the DESIGN II for Windows tutorial

Product Release Notes

The release notes for all of the versions of DESIGN II for Windows since version 2.0 are built into a Windows compiled help file installed with DESIGN II for Windows. You can access the help either through the Release Notes Icon in the DESIGN II group. You can also access the Release Notes though the Help pull down menu in DESIGN II for Windows.

Un-Installing DESIGN II for Windows

DESIGN II for Windows includes a utility for the Un-installation of all the files installed by DESIGN II for Windows. Go to the Start Button or Program Manager and select DESIGN II. The DESIGN II group icons will popup ; then select “Uninstall DESIGN II for Windows”.

Chapter 3: Basics

Starting DESIGN II for Windows

To start DESIGN II for Windows:

1. Click on the **Start** button. The Task bar displays.
2. Drag the highlight to the **Programs** folder; all folders and applications inside the Programs folder display.
3. Locate and highlight the **DESIGN II** folder. The DESIGN II for Windows program items display.
4. Drag the highlight over the **DESIGN II for Windows** item and click the left mouse button.

Exiting DESIGN II for Windows

Open the **File** menu and select **Exit**. If you have not saved your flowsheet drawing file, you will be prompted to save it.

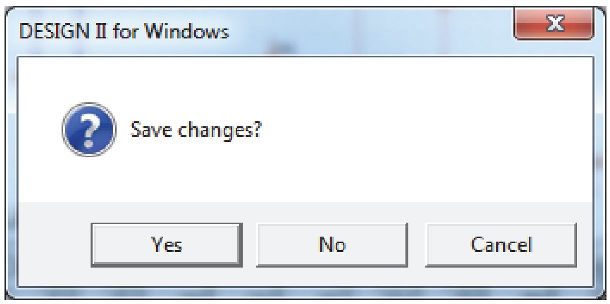


Figure 1: File Exit Dialog Dialog

Creating a New Flowsheet

Open the File menu and select New, or click on the New toolbar button. The New Drawing dialog displays.

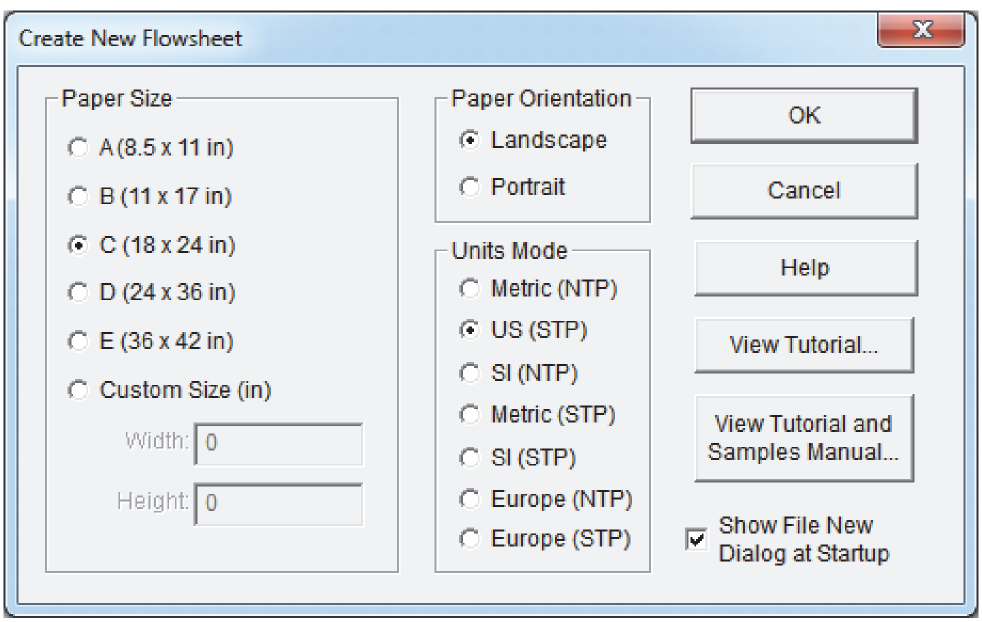


Figure 2: New Drawing Dialog

1. Click on the buttons next to the desired paper size, orientation, and mode.
2. Click the OK button. A new, blank flowsheet drawing area displays. You can now begin to create your flowsheet.

Opening an Existing Flowsheet

1. Open the **File** menu and select **Open**, or click on the **Open** toolbar button. The Open dialog displays.

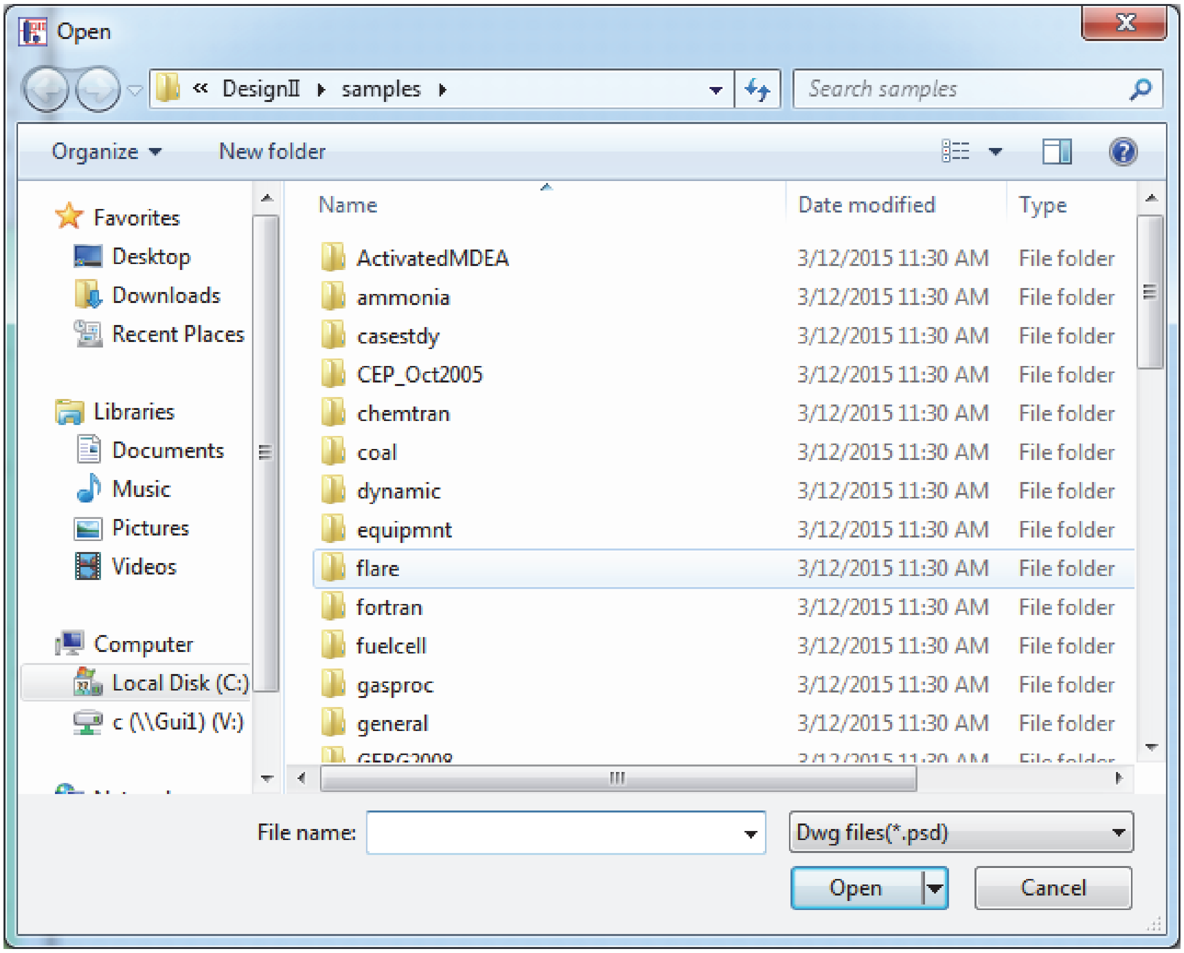


Figure 3: Open Dialog

2. Change to the desired drive and directory. Only flowsheet drawing files (files with the extension .psd) will display in the file list.

3. After locating the desired flowsheet drawing file in the list of psd files, click on its name; then click the **OK** button (or double-click on its name). The desired file opens, and you can now make any desired changes.

Printing a Flowsheet

1. Open the **File** menu and select **Print**, or click on the **Print** toolbar button. The Print dialog displays.

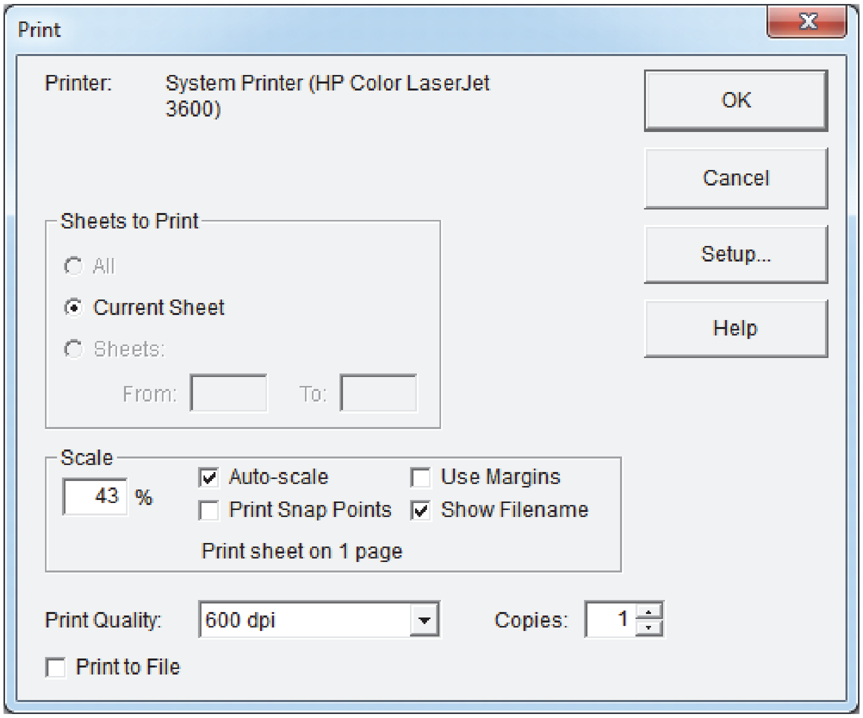


Figure 4: Print Dialog

2. Select any desired print options ; then click the **OK** button. The flowsheet is printed. The paper orientation is handled automatically; landscape (default orientation) for drawing files and portrait for text.

Setting Up a Printer

1. Open the **File** menu and select Printer Setup. If you already have the Print dialog displayed (see *Printing a flowsheet*), you can click the Setup button. The Print Setup dialog displays.

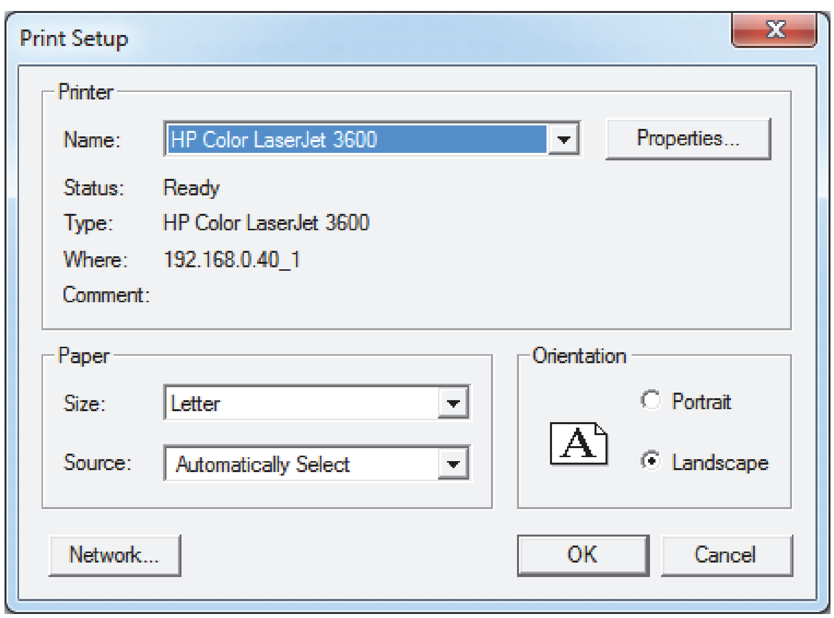


Figure 5: Print Setup Dialog

This is a standard Windows dialog. Please refer to your Windows documentation for details on completing this dialog.

2. Click **OK** when done.

Setting Print Options

See **Setting Preferences** in Chapter 4.

Saving/Naming a Flowsheet

To save a flowsheet for the first time, or create a copy of a saved flowsheet under a new name:

1. Open the **File** menu and select **Save**, or click on the **Save** toolbar button. (Select **Save As** if you want to save a copy of the flowsheet drawing file under a different name). The Save dialog displays. If you chose **Save As**, the Save As dialog displays.

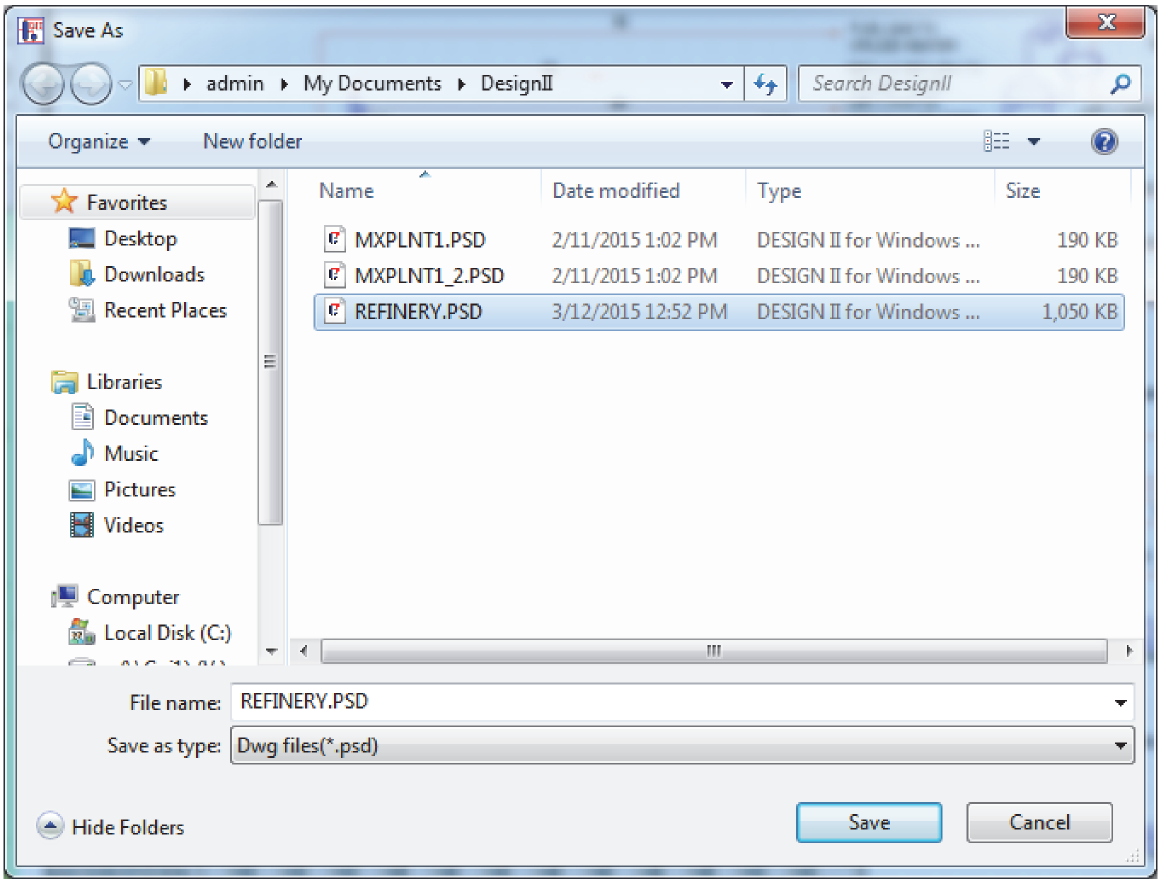


Figure 6: Save as Dialog

2. Change to the desired drive and directory where you want to save the flowsheet file.
3. Type a name for the file in the **File Name** field. (If you selected Save As, the existing file name is displayed in the File Name field; you can type over it.) The extension ***.psd*** is added to the name you type.
4. Click the **OK** button. The file is saved, and the name appears on the title bar of the DESIGN II for Windows window.

We strongly suggest you save your work periodically, especially right before you run a simulation.

To save an existing drawing (while you are working):

- Click on the **Save** button

  OR

- Hold down the **Ctrl key** and press the letter **s** key.

The drawing is saved.

DESIGN II for Windows stores the zoom and pan settings for each flowsheet; then restores them when you open the flowsheet again.

Getting Help

To view help:

1. Open the **Help** menu and select the desired help function (such as Contents, Tutorial, Procedures, etc.). If you have a dialog open, click on its **Help** button to view a topic specific to that dialog.

   The help screen displays.

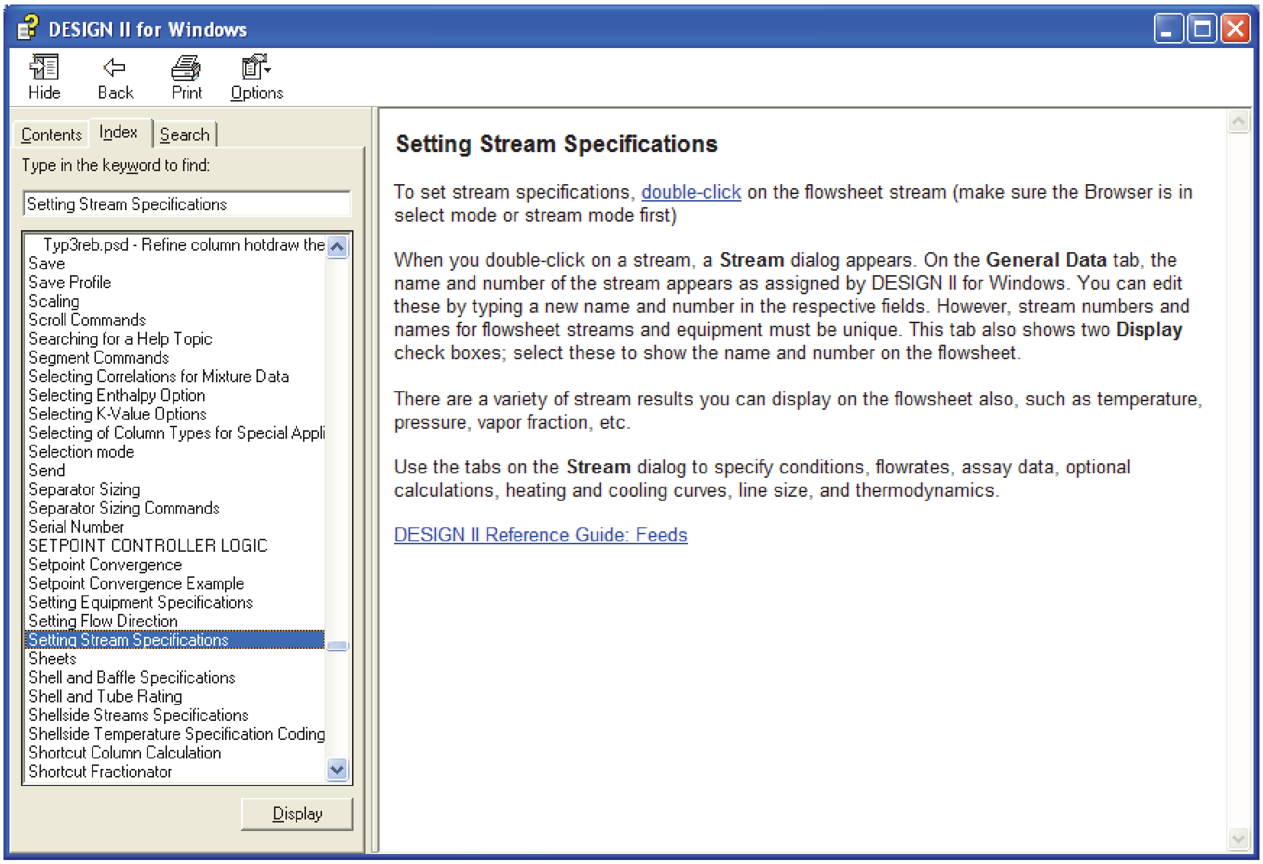


Figure 7: Example Help Screen

   Use the standard Windows procedures for using help.

2. When done, open the **File** menu and select **Exit**.

Software Issue Reporting

If DESIGN II for Windows experiences an error and the software "crashes", a software issue reporting feature allows you to report the issue to WinSim. An Internet connection is required on your PC to use this feature.

When such an issue occurs, you will see a Yes/No dialog, asking if you want to report this issue. If you click Yes; then a dialog appears asking for your name, organization, email address and a comment about the issue.

Click OK to connect to the WinSim website and email the information and the flowsheet file to customer support.

Using Edit Functions

To reduce/enlarge the size of the drawing area:

1. Open the **Edit** menu.
2. Drag the highlight over the desired option, ; then release the mouse button.

   NOTE: A checkmark next to a menu item means the item is selected.

   Ellipses (. . .) after a menu item signify a dialog box will display after you select the item.

3. You can select the following options.

Undo

This option allows you to undo the most recent operation (the operation is listed next to Undo).

Redo

This option allows you to restore the results of the most recent Undo operation (the operation to be undone is listed next to Redo).

Cut

This operation allows you to cut a selected object or objects for pasting on another area of the spreadsheet (or to another sheet or flowsheet).

Copy

This operation allows you to copy a selected object or objects for pasting on another area of the spreadsheet (or to another sheet or flowsheet). **NOTE**: All objects that you select to copy will retain the colors you set for them, when they are pasted.

Paste

This operation allows you to paste a cut or copied object. You can also copy a graphic or text from a third-party source and paste it onto your flowsheet.

Select All

This operation allows you to select all the objects on the flowsheet. **NOTE**: This is a toggle option. You must open the Edit menu and select this option again to turn off the selection of all the objects on the flowsheet. With this option turned on, you can change the color of the objects on the screen (using the Edit menu Color and Fill Color options; see below for details) or the appearance of all text (using the Text menu). If you make text changes, you will see them instantly on the flowsheet. If you make any color/fill color changes, you must turn off the Select All mode to view the changes.

Find/Find Prev/Find Next

This feature allows you to display a dialog you can use for searching. Enter the text for what you want to locate, ; then select Streams, Symbols, or Both. You can search from top to bottom (Down) or bottom to top order (Up). Click Find Next to start. If the desired object is found, the flowsheet view changes to show the item and the item is selected. The Quick List displays all of the select items (Streams, Symbols, or Both) located on the flowsheet. Click on an item from the list to select it on the flowsheet.

Select Mode/Stream Mode/Equipment Mode/Text Mode/Drawing Mode

This is a toggle that you can use to select the current selection toolbar/browser mode.

Common Stream Properties/Common Equipment Properties

If you select two or more streams or two or more equipment symbols, you can open a dialog that lets you change some common specifications that will apply to all selected streams or equipment symbols.

Properties

This operation displays the relevant dialog for a selected object on the flowsheet drawing.

Delete Item

If you have selected two or more objects on the current flowsheet drawing and want to delete them, you can use this operation. Once you delete the objects with this operation, they cannot be recovered unless you use the Undo operation.

Color

You can change the stroke (outline) color of objects and text. A standard Windows color dialog appears when you select this choice. Select the new color to use and click Okay. There are two ways to select objects/text to color: 1) either drag open a selection box to enclose the desired objects, or 2) open the Edit menu and choose Select All. You will not see any changes until the objects are de-selected (click in a blank area of the flowsheet if you used a selection box, or open the Edit menu and choose Select All to turn it off).

Fill Color

You can change the fill (interior) color of equipment. A standard Windows color dialog appears. Select the new color to use and click Okay. See the steps described for Color. You will not see any changes until the objects are de-selected.

Streams/Arrows

Select streams and arrows options, such as Style (how the line is drawn), How Many Arrows, Place Arrows Where, Arrow Size (increase the size of a selected arrow between 1-4 times its original size), and Arrow Color.

Symbols

Increase the size of a selected symbol between 1-4 times its original size. Once a size is selected, it will be used for any new symbols that are added to the drawing.

Graphics

Change the line style of a selected drawing object (e.g. rectangle, circle).

Text

Change the font size of selected text.

Legend

Displays a dialog that allows you to enter various information for use as a legend (title block) on the flowsheet.

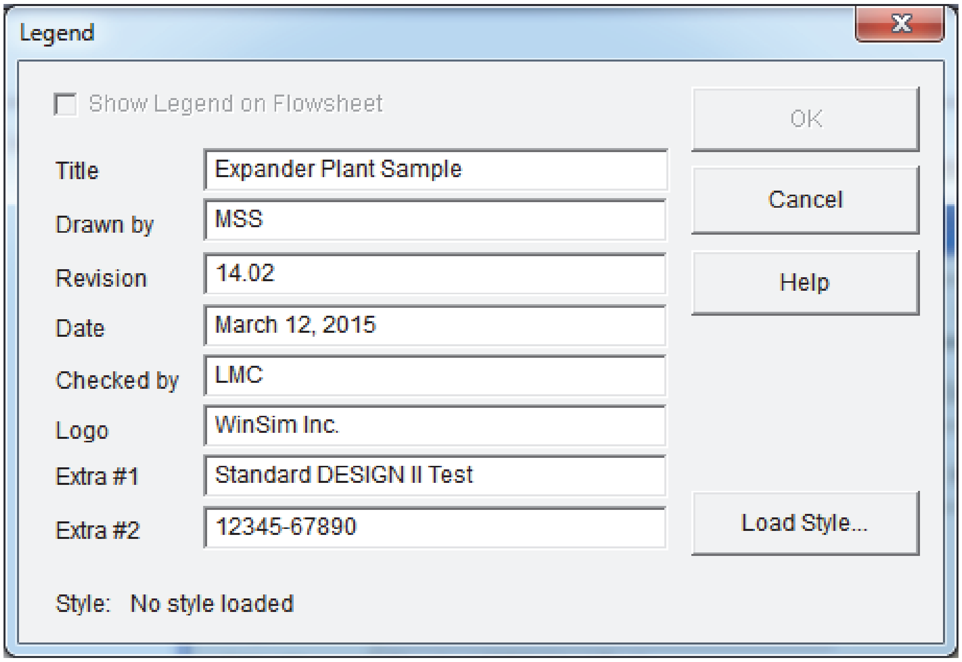


Figure 8: Legend Dialog (from expander.psd)

You can enter the following information: Title, Drawn By, Revision, Date, Checked By, Logo, and Extra #1 and 2. You can also load a pre-defined style sheet (with a .STY extension) for the legend, or use the default legend. You can then click the checkbox to show the legend on the drawing. Click OK when done. A sample legend, sample.sty is located in the installation directory, typically c:\designii.

Sheets

This operation allows you to work with the sheets that make up a flowsheet. You can add, delete or select a sheet to view.

Using View Functions

To reduce/enlarge the size of the drawing area:

1. Open the **View** menu.
2. Drag the highlight over the desired option; then release the mouse button.

   NOTE: A checkmark next to a menu item means the item is selected.

   Ellipses (. . .) after a menu item signify a dialog box will display after you select the item.
3. You can select the following options.

Redraw

Refreshes the drawing area. This is a useful option for removing extraneous pieces ("visual trash") of lines from your drawing that can result from drawing edits (this "trash" does not appear on a printed copy).

Toolbox

Toggles the Toolbox on/off on the drawing area. The Toolbox and Browser essentially provide the same tools.

Browser

Toggles the Browser on/off on the drawing area.

Toolbar

Toggles the Toolbar (the bar with the icons, below the menus) on/off on the drawing area.

Status Bar

Toggles the Status Bar (the bar at the bottom of the window that displays messages) on/off on the drawing area.

Zoom Area

You can click on the Zoom Area toolbar button. This function allows you to select a desired area to zoom. The cursor changes to a magnifying glass; drag open a dashed-line box to enclose all of the drawing area that you want to zoom. When you release the mouse button, the drawing zooms in.

Zoom In

You can also click on the Zoom In toolbar button. This function enlarges the drawing using a pre-set zoom amount.

Zoom Out

You can also click on the Zoom Out area toolbar button. This function reduces the drawing using a pre-set zoom amount. Once you have reached the maximum zoom in or out amount, you can no longer zoom in that direction.

Actual Size

You can also click on the Actual Size toolbar button, or hold down the Ctrl key and press the 1 key. This function changes the drawing to its actual size (a 1:1 ratio).

Full Page

You can also click on the Full Page toolbar button, or hold down the Ctrl key and press the key This operation displays the entire drawing, scaling it to occupy the current program window size. As you change the window size, the drawing scales appropriately.

Note: DESIGN II for Windows will save the zoom and pan (current view of the flowsheet within in the window) automatically, so when you open the flowsheet again these settings are restored.

Rulers

Displays or hides the Vertical/Horizontal Rulers. The cursor position is indicated by a dashed line over the respective ruler. The distance between division marks on the ruler changes when you change the scale of the drawing. This distance matches very closely that of the printed flowsheet when you select the option Actual Size under the View menu.

Set Rulers

Displays a dialog, which you can use to set the unit of measurement for the ruler (either inches or centimeters).

A slider bar allows you to set the width of the ruler on-screen; drag the box on the slider to the left to make the rulers more narrow or to the right to make them wider. Click OK when done.

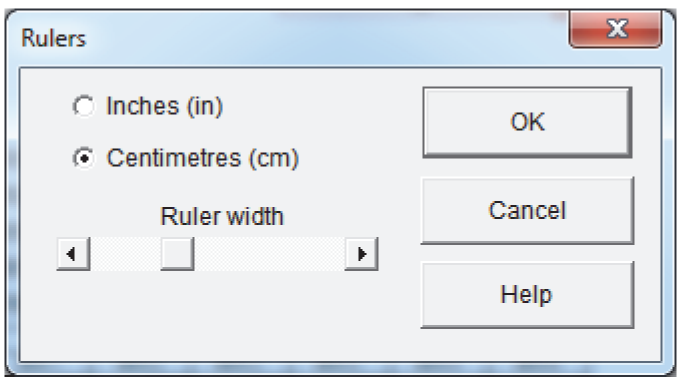


Figure 9: Rulers Dialog (from expander.psd)

Grid/Dense Grid

This operation toggles on/off the display of a grid on the drawing area. The grid provides convenient reference points for positioning equipment and streams on your flowsheet. Equipment symbols are also scaled in terms of grid units and are designed to have their boundaries fall on grid points. The grid does not appear when you print the drawing. If you have the grid turned on and would like more a grid with more points closer together, select Dense Grid.

General Simulation Results

This displays the General Simulation Results window. Until you have run a simulation, this window remains blank.

Results

This displays the Equipment/Stream Results window for the selected equipment or stream.

Working With Program Components

Window components

The DESIGN II for Windows screen is made up of the following components:

- Title bar/window frame/window control boxes
- Menu bar
- Tool bar

- Toolbox (or Browser, depending on what you have selected under the View menu()
- Drawing area
- Status line

Refer to the graphic on the next page showing you where these components are located on-screen.

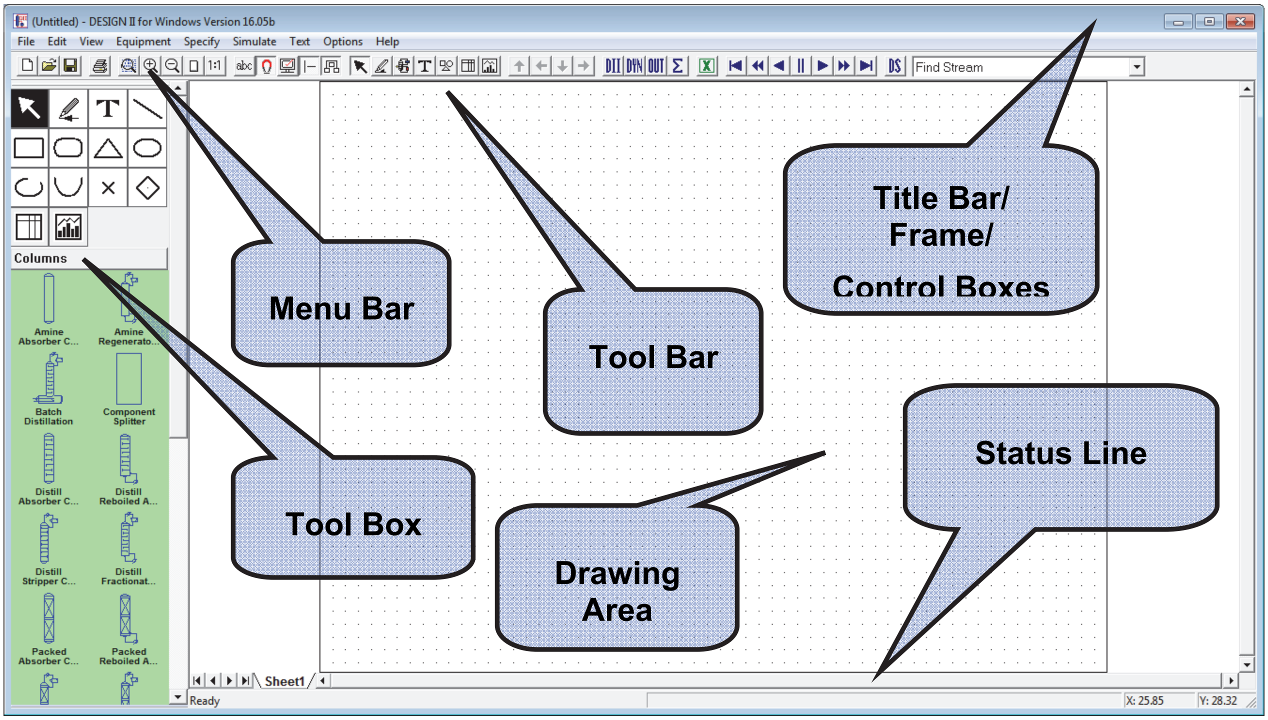


Figure 10: DESIGN II for Windows Screen and Components

Title bar/window frame/window control boxes

These are standard windows components. Use them as you normally would with other Windows applications. For details, refer to your Windows User documentation.

Menu bar

To open a main menu item, click on its name. Its menu displays. To select an item from the menu, drag the highlight over it and click.

Another option is to click on a main menu item; then drag the highlight over the desired item and release the mouse button.

A menu shortcut uses the keyboard. Each main menu item has an underlined letter in its name. To open a menu, hold down the Alt key and press the key corresponding to the underlined letter. For example, the File menu has the letter F underlined. To open the File menu, hold down the Alt key and press the letter f key (this is not case sensitive, so you do not need to hold down the Shift key). The menu opens.

Some menu items produce a sub menu, called a slide right menu. If a menu item has an associated slide right menu, a black arrowhead appears next to the item, pointing to the right. To open a slide right menu, highlight the menu item; the slide right menu displays. Move the cursor over the slide right menu, ; then drag the highlight over the desired item and click.

Tool bar

The toolbar displays commonly used DESIGN II for Windows functions, represented as buttons. Each button also has an equivalent item listed on a menu, or a Browser function. Click on the desired button to perform the function.

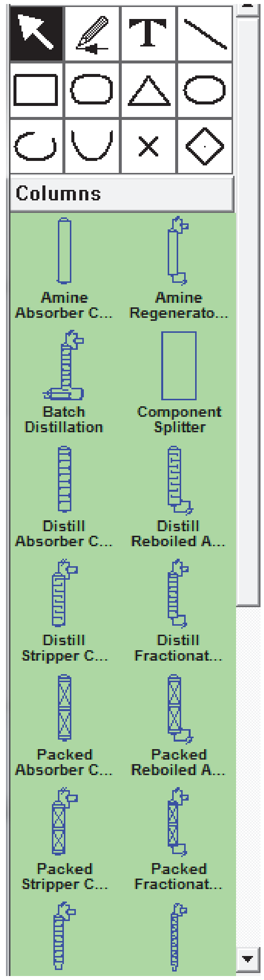
Drawing area

This is where you will draw your flowsheet. You can turn on a grid for aligning objects (see the Options/Preferences chapter for details).

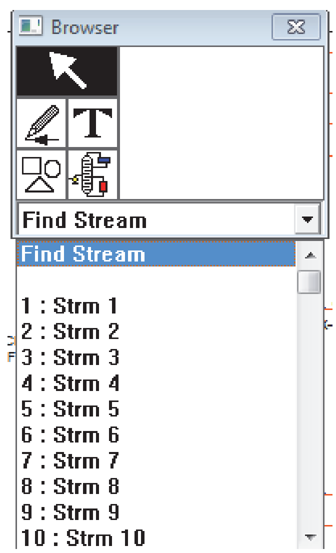
Toolbox (or Browser)

To use the Toolbox:

Click on the desired Toolbox button (Selection, Stream, Text, Drawing Elements, Stream, and Equipment). The Toolbox shows the selected tool.

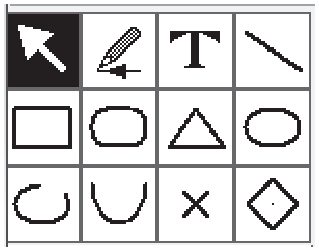


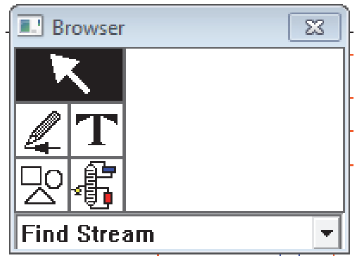
If you are using the Browser: the drop down list on the bottom of the Browser changes based on the tool selected. For example, if you choose the Selection Tool you will see streams, equipment, and other objects on the flowsheet:



If you select the Selection tool:

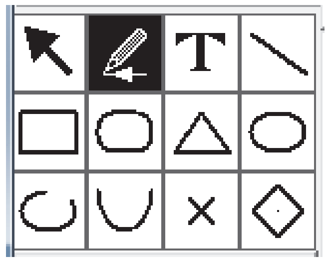
You can select individual streams, arrows, text and equipments. Or, you can select groups of items in the flowsheet for copying, deletion or moving.

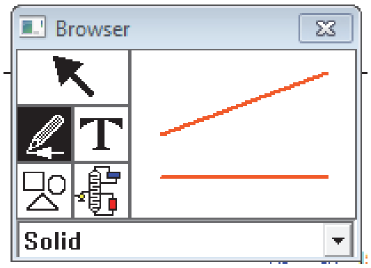




If you select the Stream tool:

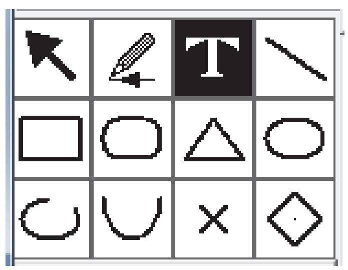
You can draw (indicated by a diagonal and straight line). Use the Edit menu’s Streams/Arrows options to change line style, number of arrows, etc. **Browser only**– The drop down list allows you to select the line style.

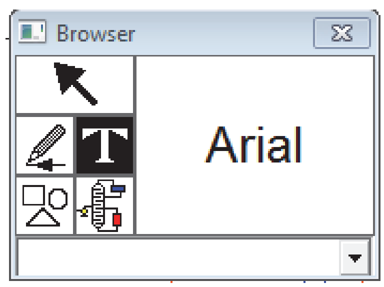




If you select the Text tool:

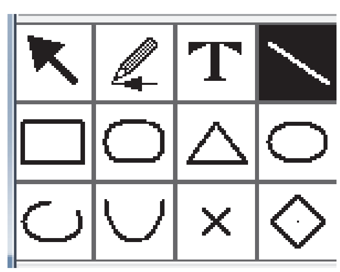
You can add text to your flowsheet. Additional fonts and font sizes are available by clicking on the Text option from the menu bar. **Browser only**– The Browser Display shows you will draw text (indicated by the selected font name). The drop down list allows you to select the size of the text you type.





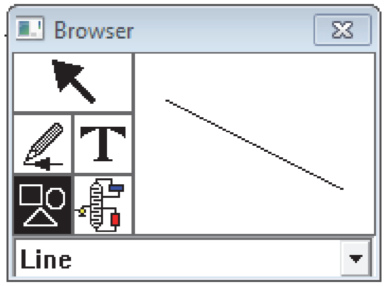
Toolbox only– If you select a drawing element tool:

You can add the drawing element (line, rectangle, circle, etc.) to your flowsheet.



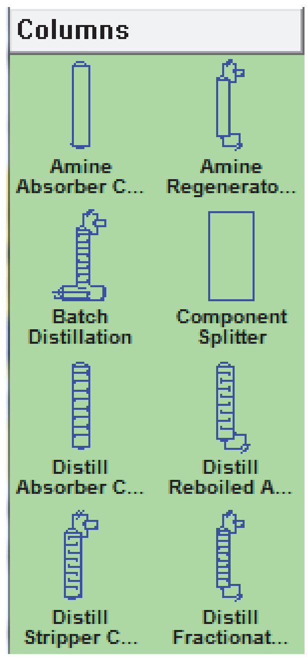
Browser only– If you select the Drawing Elements tool:

The Browser Display shows that you can choose to draw a line, rectangle, rounded rectangle, triangle, ellipse, arc parabola, or a snap point (for use when creating custom symbols).



Toolbox only– If you select an equipment type:

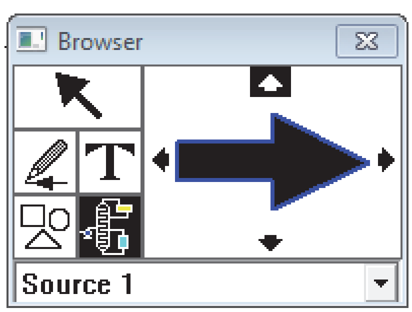
You can add the equipment to your flowsheet. The equipment symbols are grouped by type. Click on the name of an equipment type to show/hide the corresponding symbols. The equipment types are: Columns, Heat Exchangers, Pressure Change, Reactors, Stream Operations, Controls and Metering, Tanks, and Miscellaneous. If you have created a custom library, you can choose a symbol from it.



Browser only– If you select the Equipment tool:

The equipment symbols are grouped by type on the drop down list (see the list of types in the previous Toolbox equipment type description). The Browser Display shows the selected Equipment Symbol you will draw; four direction arrows also display. Click on a direction arrow to change the orientation of the Equipment Symbols you draw. The Equipment Symbol in the display re-orients to the newly selected direction.

The drop down list allows you to select the desired Equipment Symbol.



Status line

View the drawing status (Ready) and the X/Y coordinates of the cursor position in the Drawing area.

Tool tips

A tooltip with relevant information will appear for a stream or equipment symbol when hovering the mouse over it.

Working with Dialogs

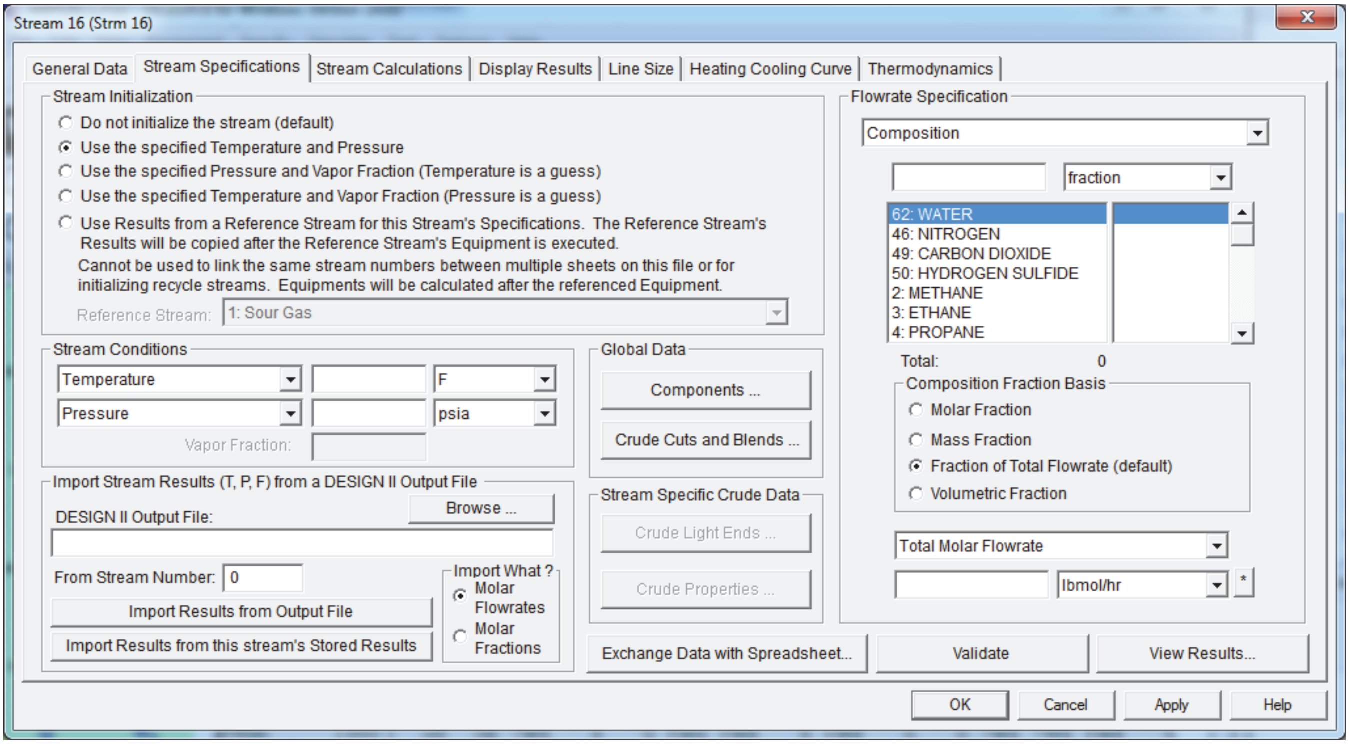


Figure 11: Example Dialog (Stream) (from MxpInt1.psd)

Work with dialogs as you normally would in Windows (depending on which type of Windows software you are using on your PC). The OK button closes the dialog and acts upon the information you entered into the dialog. The Cancel button closes the dialog and does not act upon the information you entered in the dialog. The Apply button saves your changes but leaves the dialog open. If there is a Help button, it displays a help topic specific to the current dialog.

If you select two or more streams or two or more equipment symbols, you can view a common properties dialog. After selecting the objects, open the **Edit** menu and select **Common Stream Properties** or **Common Equipment Properties**, as appropriate. You will see the relevant Common Properties dialog for viewing/setting common display results and thermodyanic options.

Using Multiple Sheets in a Flowsheet

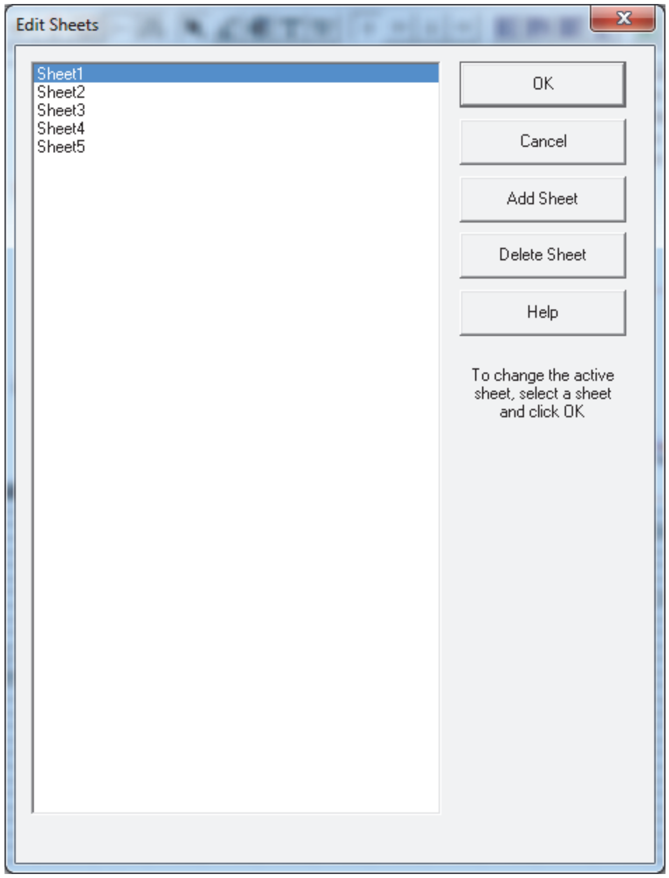


Figure 12: Edit Sheets Dialog (from links.psd sample)

You can have one or more sheets in your flowsheet. The sheets will be automatically named “Sheet” plus their tab number. New Sheets will be added after the last sheet in the set. You can display this dialog by opening the **Edit** menu and selecting Sheets or by right clicking on any of the sheet tabs and selecting Properties. You can also rename or delete a sheet by right clicking on the desired sheet tab.

Please note that the first sheet cannot be deleted. And, when you calculate the flowsheet, all information on all the sheets will be used for the simulation.

If you have streams that start on one sheet and terminate on another sheet then you will need to indicate which stream(s) are connected to that primary stream by clicking the check box on the General Data tab on the Stream properties dialog of the second occurance of the stream on the second sheet. The first occurance of the stream on the first sheet will not have its box checked since it is the “primary” occurance of the stream.

You can disable (and enable) any sheet in the flowsheet. You can also disable streams and equipment. Please note that all validation rules apply as if the disabled stream, equipment or sheet does not exist. For instance, if anequipment is disabled then all product streams from that equipment must now be specified as if they are feed streams to the process. Disabled sheets will have "Disabled" written across the sheet and an X placed across the sheet. Right click on the sheet and select Enable or Disable from the pop-up menu.

Exchange Data with Spreadsheet

You can import specifications from Excel before a simulation and export calculated results to Excel after a simulation. You can set the actual data transfer items to use. Currently, only compressors, line, one stream heat exchangers/two stream heat exchangers, expanders, air coolers, dividers, multiple dividers, distillation columns, flash separators, and streams are supported; other unit modules will be added in the future.

Open the **dialog** for the desired equipment or stream and click on the Exchange Data with Spreadsheet button.

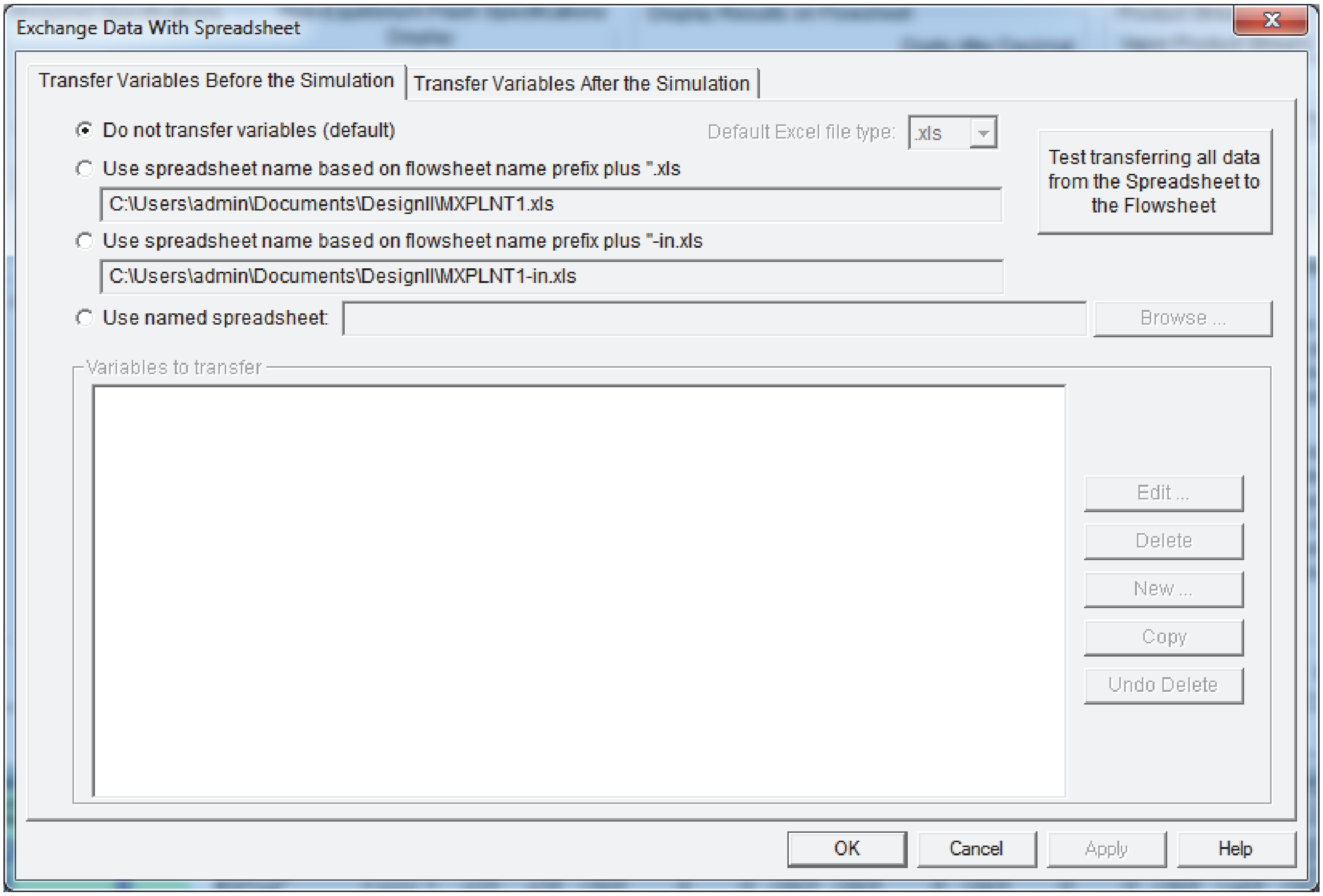


Figure 13: Exchange Data with Spreadsheet Dialog: Transfer Variables before the Simulation (from Mxplnt1.psd)

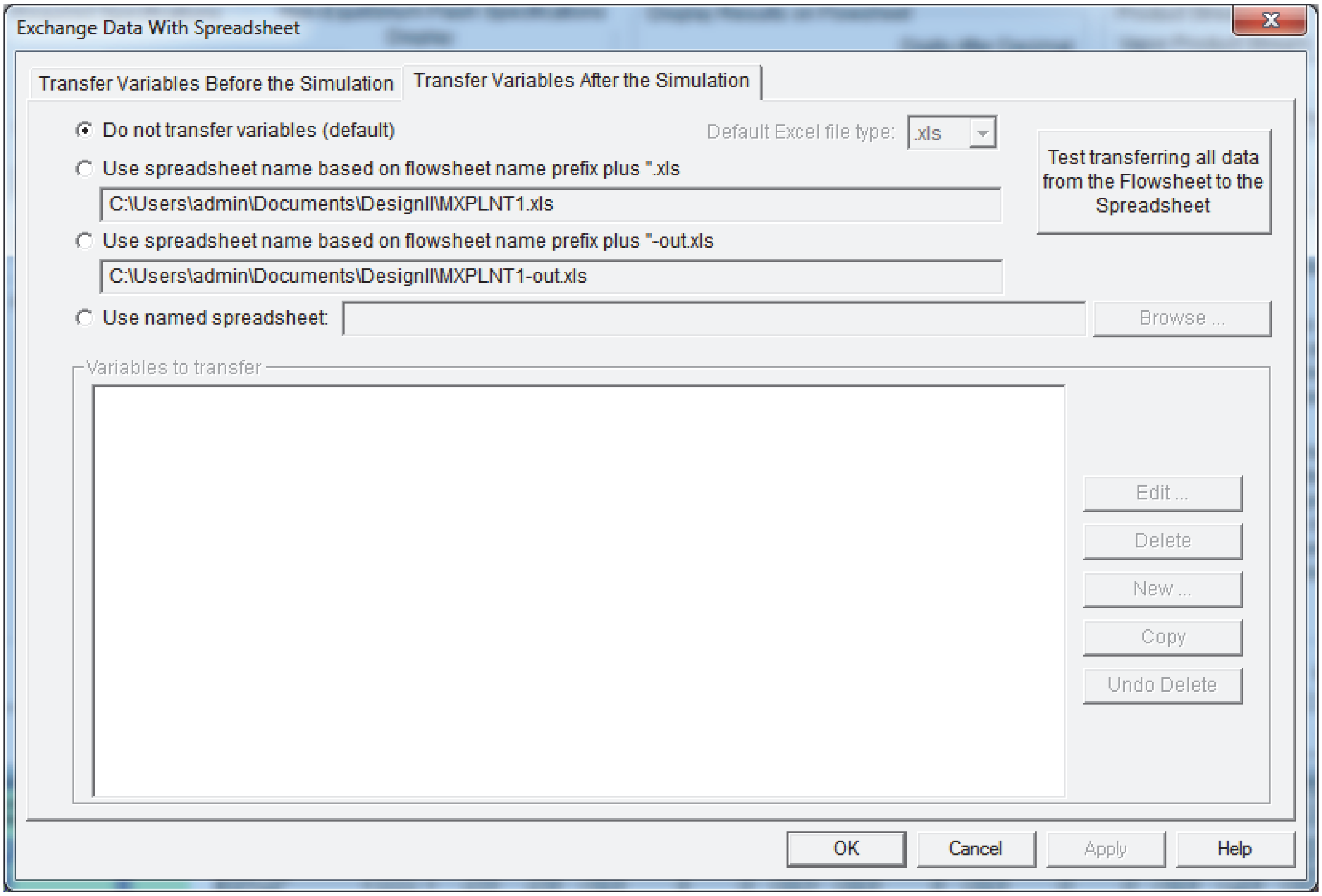


Figure 14: Exchange Data with Spreadsheet Dialog: Transfer Variables After the Simulation (from Mxplnt1.psd)

Transfer Variables Before the Simulation

Select one of the following choices:

Do not transfer variables (default)

Use spreadsheet name based on flowsheet name prefix plus ".xls": this shows the path where the file must be located, along with the expected filename.

Use spreadsheet name based on flowsheet name prefix plus "-in.xls": this shows the path where the file must be located, along with the expected filename.

Use named spreadsheet: click the Browse button to display a dialog for choosing the drive/directory and file name. When you select the file and click Open on the dialog, the name appears in the Use named spreadsheet field.

Variables to transfer: You can use this option to select specific variables in the spreadsheet to transfer into the flowsheet. Select New to display the Transfer a Variable from a Flowsheet dialog for selecting variables. To edit an existing variable, click on its name in the list and click the Edit button. To remove a variable, click on its name then click Delete. To duplicate a variable so you can go in and make minor edits to create a new variable, click on its name in the list and click Copy.

Transfer Variables After the Simulation

Select one of the following choices:

Do not transfer variables (default)

Use spreadsheet name based on flowsheet name prefix plus ".xls": this shows the path where the file will be located, along with the projected filename.

Use spreadsheet name based on flowsheet name prefix plus "-out.xls": this shows the path where the file will be located, along with the projected filename.

Use named spreadsheet: click the Browse button to display a dialog for choosing the drive/directory and file name. When you select the file and click Open on the dialog, the name appears in the Use named spreadsheet field.

Variables to transfer: You can use this option to select specific variables in the flowsheet to transfer into the spreadsheet. Select New to display the Transfer a Variable from a Flowsheet dialog for selecting variables.

Test transferring all data from the Spreadsheet to the Flowsheet/Test transferring all data from the Flowsheet to the Spreadsheet

Once you've set up a variable transfer (either from the spreadsheet or from the flowsheet), click the appropriate button to test the data transfer before you actually perform the final data transfer.

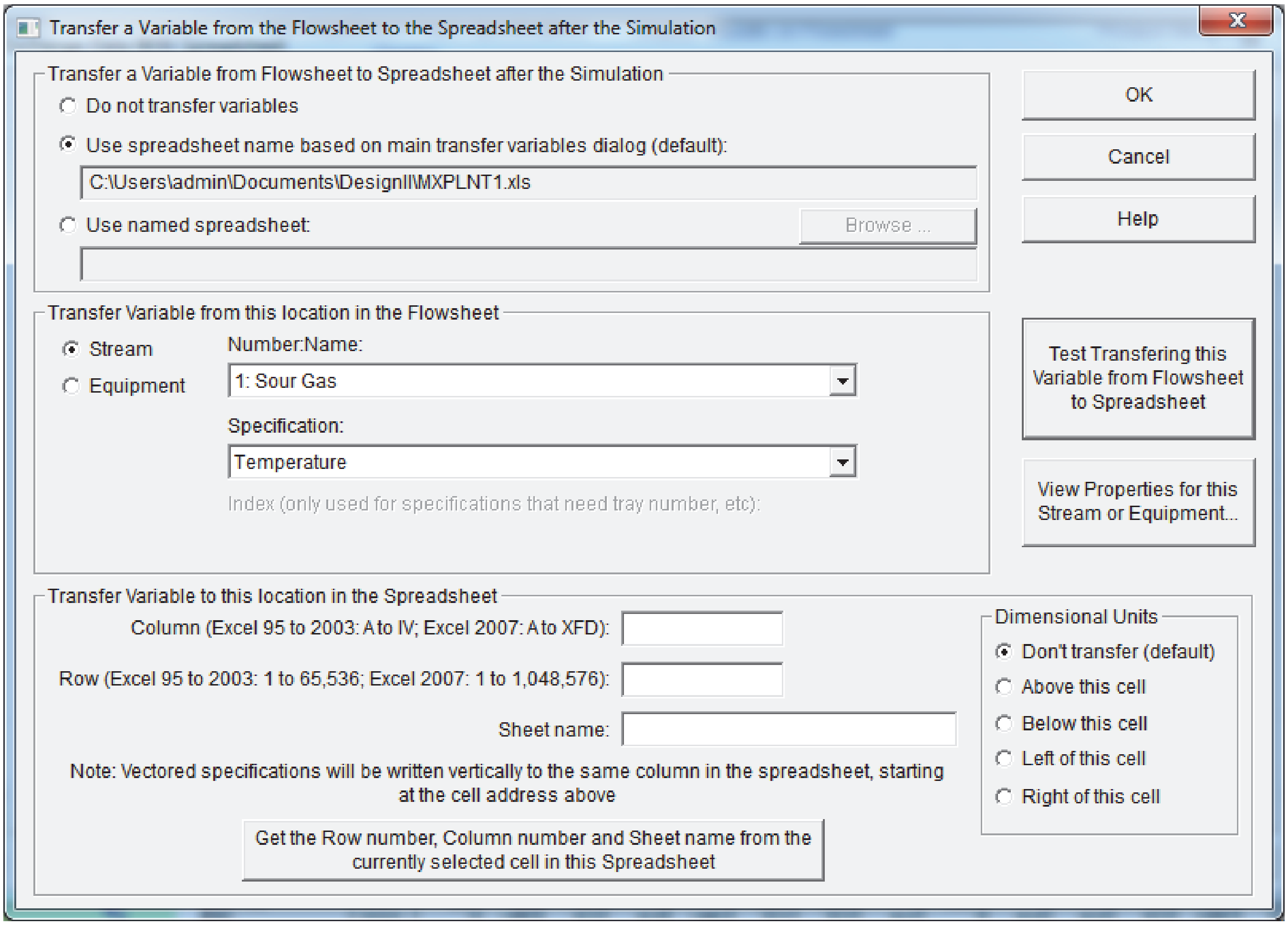


Figure 15: Transfer a Variable from a Flowsheet to the Spreadsheet after the Simulation (from MxpInt1.psd)

Transfer a Variable from the Flowsheet to Spreadsheet After the Simulation

Select one of the following choices:

Do not transfer variables

Use spreadsheet name based on main transfer variables dialog (default) this shows the path where the file will be located, along with the projected filename.

Use named spreadsheet: click the Browse button to display a dialog for choosing the drive/directory and file name. When you select the file and click Open on the dialog, the name appears in the Use named spreadsheet field.

Transfer Variable from this location in the

Select Stream or Equipment.

Number:Name: Select the desired object (either stream name or equipment name) from the list. For

Spreadsheet

equipment, you can only select a heat exchanger or air cooler.

Specification: After selecting the desired object, open the list and select the specification you want to transfer to Excel.

NOTE: Vectored specifications will be written vertically to the same column in the spreadsheet, starting with the cell address (column, row, sheet) that you will enter below.

Transfer Variable to this location in the Spreadsheet

Column (Excel 95 to 2003: A to IV; Excel 2007: A to XFD): based on the version of Excel you are using, enter the appropriate column letter to which you want to write the variable.

Row (Excel 95 to 2003: 1 to 65,536; Excel 2007: 1 to 1,048,576): enter the appropriate row number to which you want to write the variable.

Sheet name: Enter the name of the worksheet to which you want to write the variable.

Get the Row number, Column number, and Sheet name from the currently selected cell in this Spreadsheet: if you have the desired spreadsheet file open in Excel, click on the variable cell; then click this button to automatically populate the Column, Row, and Sheet name fields on this dialog.

Dimensional Units: select how to handle the transfer of dimensional units for the variable data to the spreadsheet based on the column/row/sheet name you entered:

Don't transfer (default)

Above this cell

Below this cell

Left of this cell

Right of this cell

Test Transferring this Variable from Flowsheet to Spreadsheet

Once you've set up a variable transfer, click this button to test the variable data transfer before you actually perform the final transfer.

View Properties for this Stream or Equipment

Once you've selected the desired object (stream or equipment), click this button to open the corresponding properties dialog.

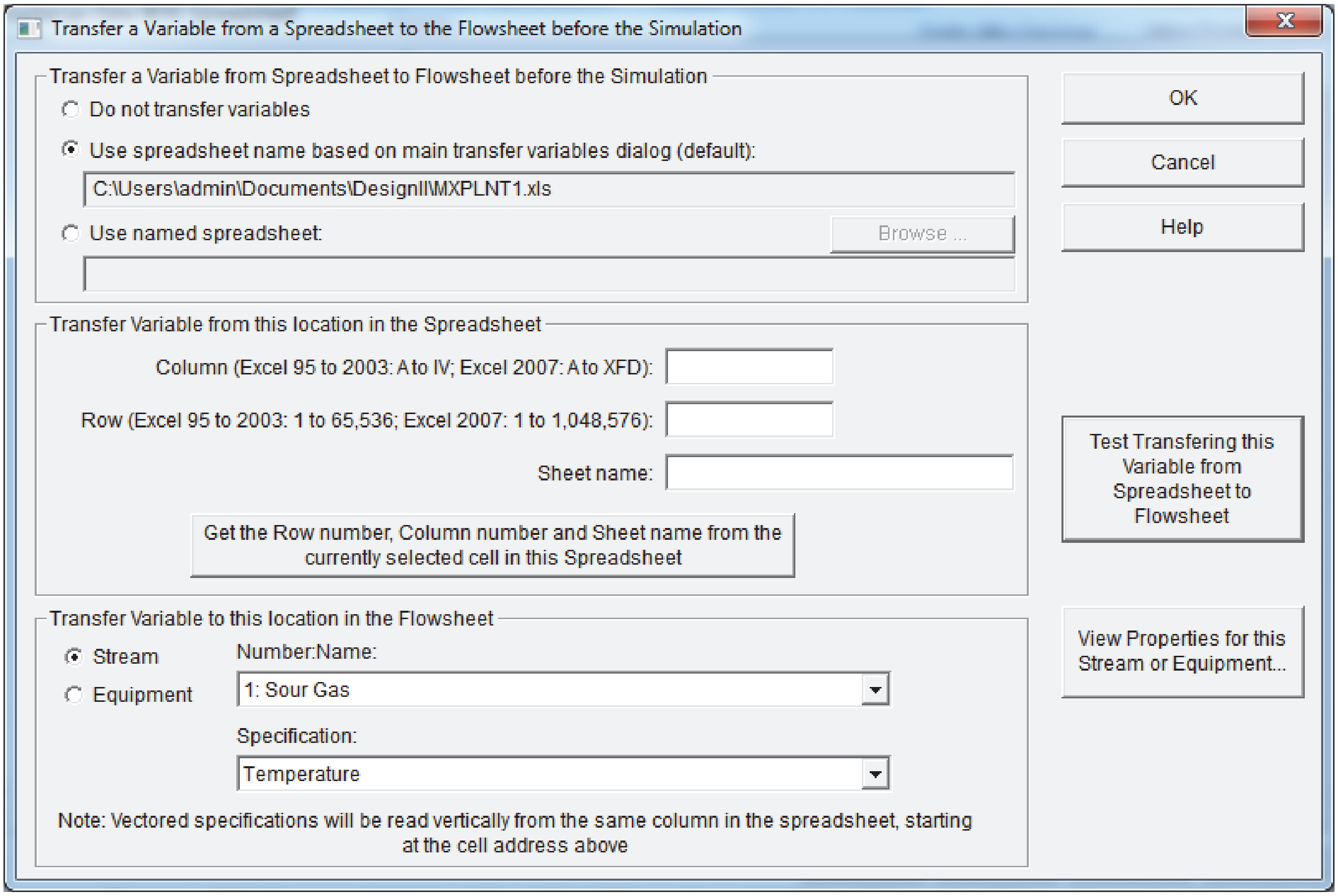


Figure 16: Transfer a Variable from a Spreadsheet to the Flowsheet before the Simulation (from Mxplnt1.psd)

Transfer a Variable from Spreadsheet to Flowsheet before the Simulation

Select one of the following choices:

Do not transfer variables

Use spreadsheet name based on main transfer variables dialog (default) this shows the path where the file must be located, along with the expected filename.

Use named spreadsheet: click the Browse button to display a dialog for choosing the drive/directory and file name. When you select the file and click Open on the dialog, the name appears in the Use named spreadsheet field.

Transfer Variable from this location in the Spreadsheet

Column (Excel 95 to 2003: A to IV; Excel 2007: A to XFD): based on the version of Excel you are using, enter the appropriate column letter that contains the variable.

Row (Excel 95 to 2003: 1 to 65,536; Excel 2007: 1 to 1,048,576): enter the appropriate row number that contains the variable.

Sheet name: Enter the name of the worksheet that contains the variable.

Get the Row number, Column number, and Sheet name from the currently selected cell in this Spreadsheet: if you have the desired spreadsheet file open in Excel, click on the variable cell; then click this button to automatically populate the Column, Row, and Sheet name fields on this dialog.

Transfer Variable to this location in the Flowsheet

Select Stream or Equipment.

Number/Name: Select the desired object (either stream name or equipment name) from the list. For equipment, you can only select a heat exchanger or air cooler.

Specification: After selecting the desired object, open the list and select the specification to which you want to transfer the variable from Excel.

NOTE: Vectored specifications will be read vertically from the same column in the spreadsheet, starting with the cell address (column, row, sheet) that you entered above.

Test Transferring the Variable from Spreadsheet to Flowsheet Once you've set up a variable transfer, click the appropriate button to test the variable data transfer before you actually perform the final transfer.

View Properties for this Stream or Equipment Once you've selected the desired object (stream or equipment), click this button to open the corresponding properties dialog.

Send Results to Spreadsheet

After you have generated results (i.e. run a simulation) you can click this button on an Equipment or Stream dialog to open up Excel and place related data into a spreadsheet.

The data exported depends on the item for which you choose to send the results to the spreadsheet.

The spreadsheet file is named using the item name (such as the type of equipment and corresponding number).

You can then use the spreadsheet however you want to communicate the resulting data.

Validate

You can use this button to check for errors on the current object (e.g. equipment, stream). If no errors are found, a message states that no validation errors were detected. If any errors are found; then you are instructed on what to correct.

View Results

After you have generated results (i.e. run a simulation), you can click this button on an Equipment or Stream dialog to display a window that allows you to view the results for that specific piece of equipment or stream.

Comments

You can enter optional notes for the selected object (equipment or stream).

Export Current Sheet to DXF

You can export the current flowsheet in a file format that can be imported into AutoCAD®.

Open the **File** menu and select **Export Current Sheet to DXF file**. When you select this choice, a Save As dialog appears so you can select a desired drive and directory for the file. The name will use the current flowsheet name with the extension .dxf added to it. You can then import the file into AutoCAD. All the drawing elements will be exported (symbols, streams, text blocks, etc.)

Chapter 4: Options / Preferences

Setting Preferences

You can set your preferences by:

1. Open the **Specify** menu and select **Preferences**. The ***General Preferences*** dialog displays.

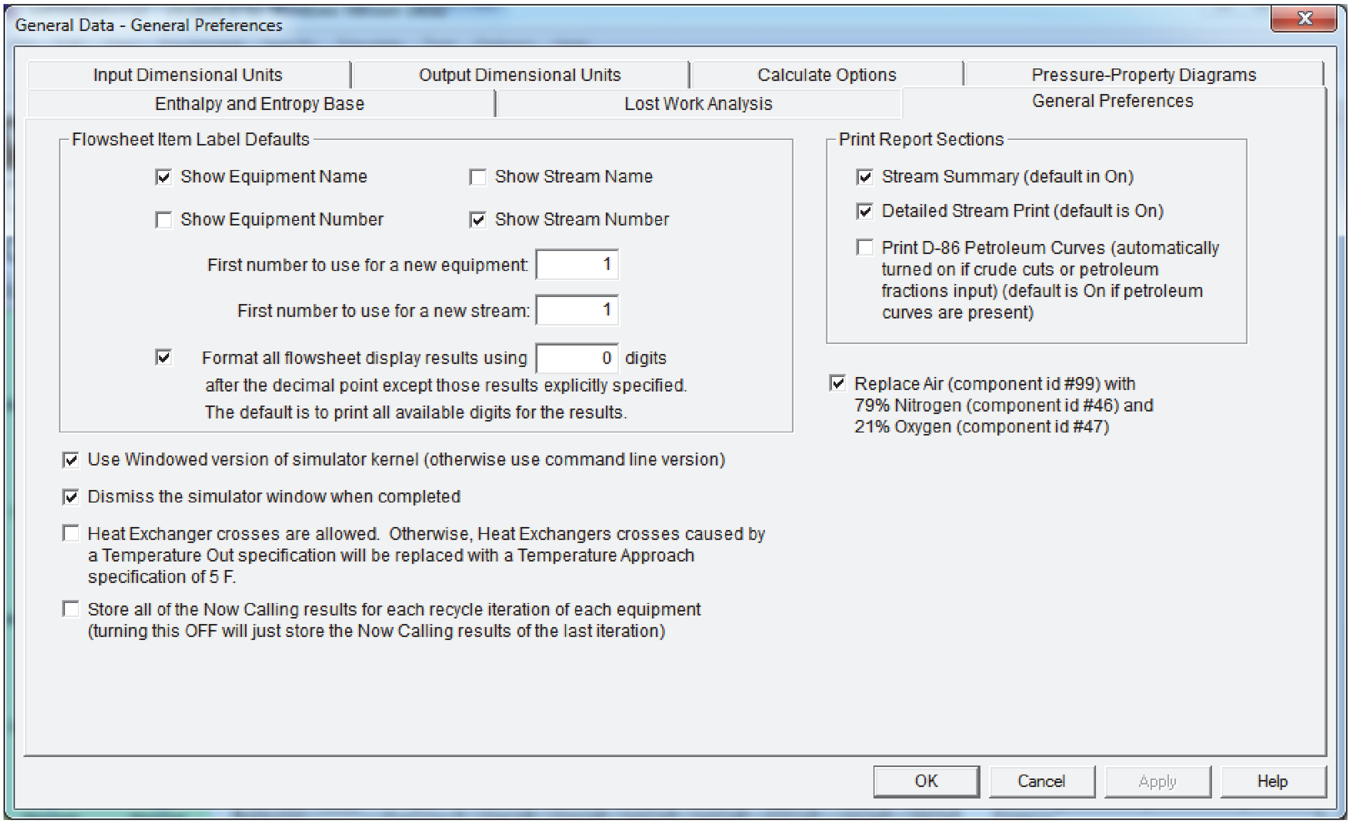


Figure 1: General Preferences Dialog (from expander.psd)

2. Select the desired preferences, as described below.

Flowsheet Item Labels
: This list of check boxes allows you to choose the labels displayed on the flowsheet you construct. You can select any combination of these options.

Show Equipment Name
: Display the equipment name on the flowsheet drawing.

Show Equipment Number
: Display the equipment number on the flowsheet drawing.

Show Stream Name
: Display the stream name on the flowsheet drawing.

Show Stream Number
: Display stream numbers on the flowsheet drawing.

Use Windowed Version of Simulator Kernel
: Click this **checkbox** to use the 32-bit Windows version of the simulator program for Windows; if it is not checked, the command line version is used..

Dismiss Simulator Window When Complete
: Click this **checkbox** to dismiss the simulator window automatically when the simulation is complete.

Heat Exchanger crosses are allowed. Otherwise, Heat Exchanger crosses caused by a Temperature To Specification will be replaced with a Temperature Approach specification of 5 F
: Click this check box to permit Heat Exchanger crosses in the simulation.

Store all of the Now Calling results for each recycle iteration of each equipment (turning this OFF will just store the Now Calling results of the last iteration)

When using the recycle function, click this checkbox to store the Now Calling results generated during each iteration for the equipment/streams on the flowsheet. If you do not check it, only the Now Calling results generated during the last iteration are stored.

3. Click the **OK** button when done. The Preferences dialog closes and your selected preferences are set.

Input Dimensional Units

Use this tab to select input dimensional units for DESIGN II and DESIGN II for Windows.

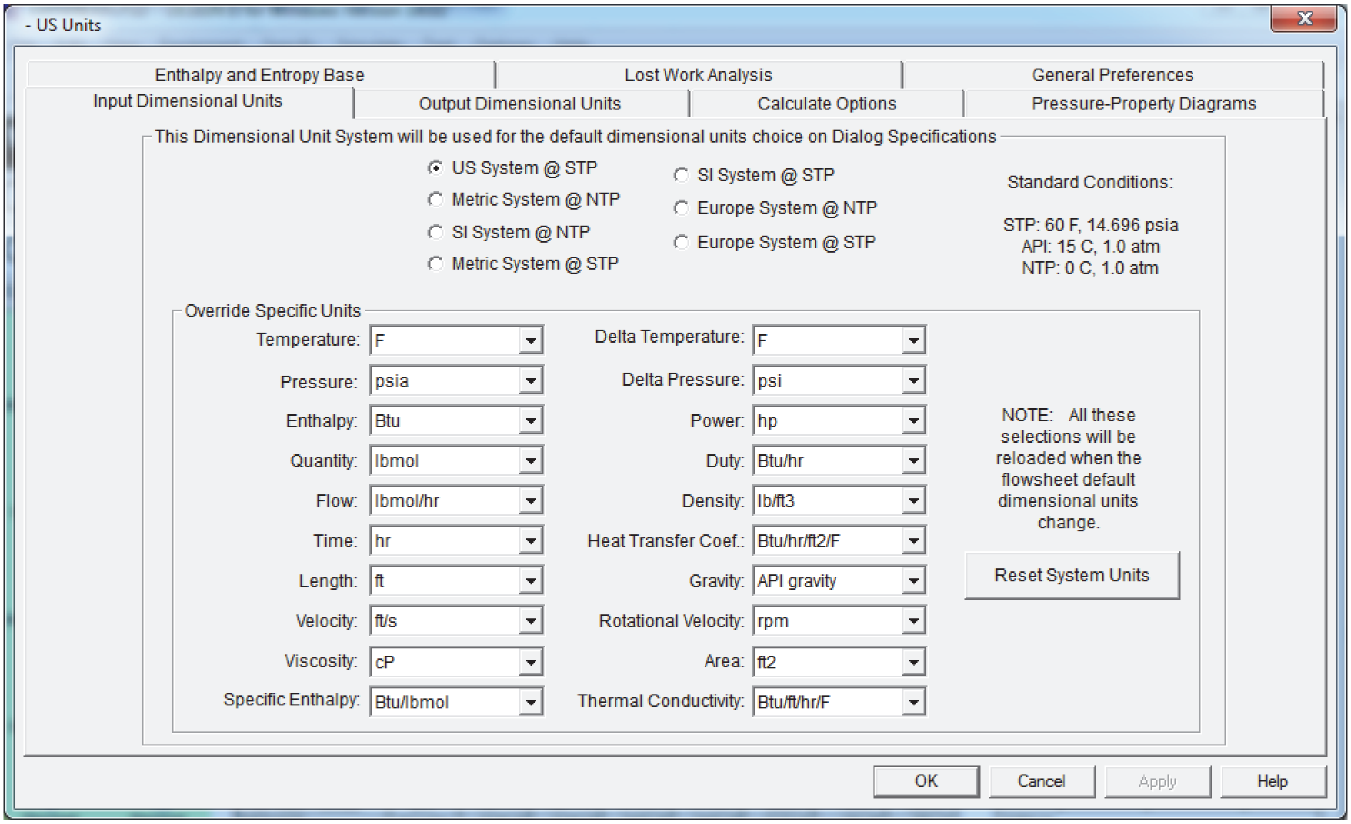


Figure 2: Input Dimensional Units Preference Dialog (from expander.psd)

US System @ STP

Output units will be in the US system of units (pounds, feet, Fahrenheit, etc.) at STP.

Metric System @ NTP

Output units will be in the Metric system of units (kilograms, meters, centigrade, etc.) at NTP.

SI System @ NTP

Output units will be in the SI system of units (kilograms, meters, Kelvin, etc.) at NTP.

Metric System @ STP

Output units will be in the Metric system of units (kilograms, meters, centigrade, etc.) at STP.

SI System @ STP

Output units will be in the SI system of units (kilograms, meters, Kelvin, etc.) at STP.

Override Specific Units

Use this section to select the different types of dimensional units that are used for DESIGN II input. To change an input unit, open the corresponding unit list and select the type of unit.

Any of your selections will be reloaded when the flowsheet dimensional units change.

To reset the units to the default units, select the **Reset System Units** button.

Output Dimensional Units

Use this tab to select the dimensional units to use for simulation results.

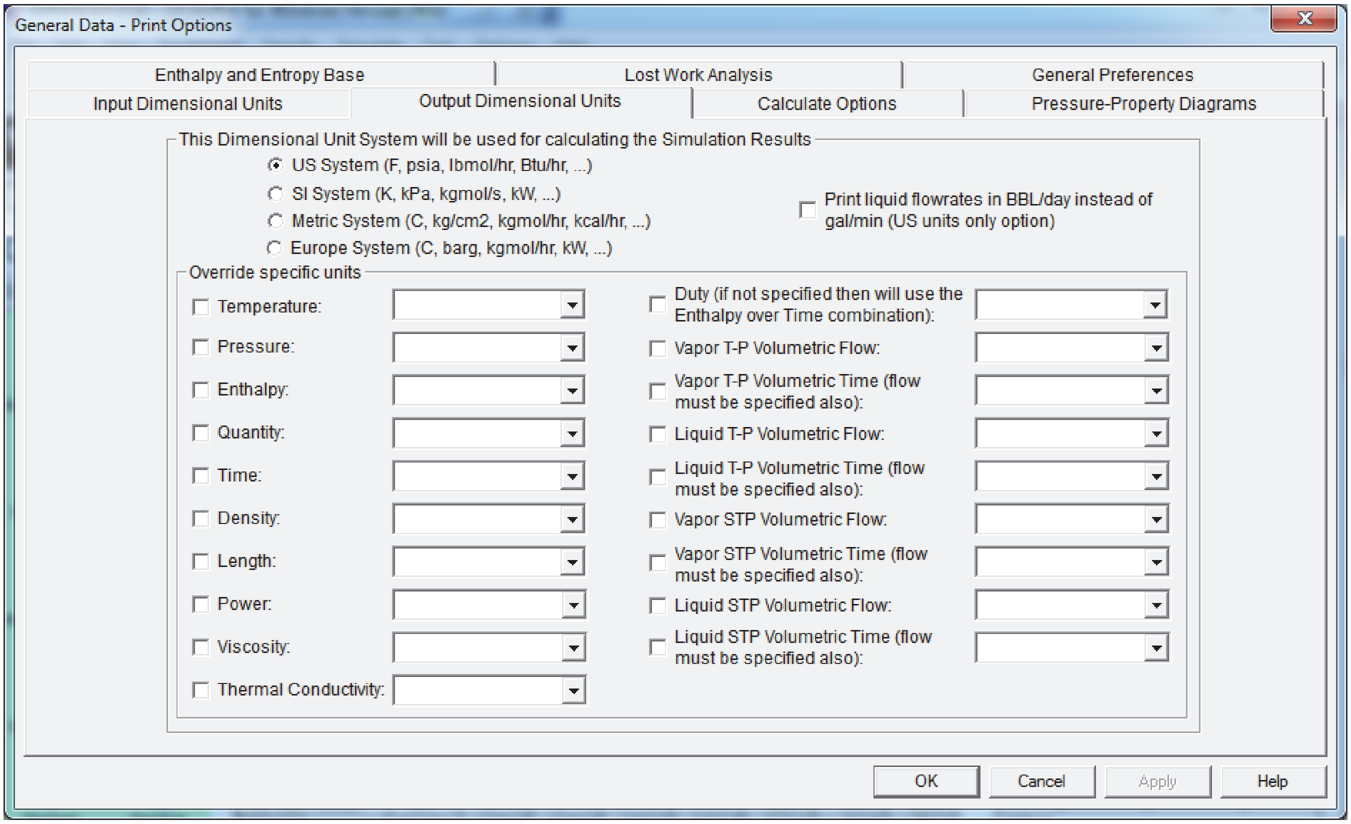


Figure 3: Output Dimensional Units Preference Dialog (from expander.psd)

US System (F, psia, lbmol/hr, Btu/hr, ...)
: Output units will be in the US system of units (pounds, feet, Fahrenheit, etc.)

SI System (K, kPa, kgmol/s, kW, ...)
: Output units will be in the SI system of units (kilograms, meters, Kelvin, etc.)

Metric System (C, kg/cm2, kgmol/hr, kcal/hr, ...)
: Output units will be in the Metric system of units (kilograms, meters, centigrade, etc.)

Print liquid flowrates in BBL/day instead of gal/min (US units only option)
: If you use select US System, select this checkbox if you want to display liquid flowrates on the output as BBL/day.

Override Specific Units
: Use this section to select the different types of dimensional units that are used for DESIGN II output. To change an output unit, open the corresponding unit list and select the type of unit.

Calculate Options

Use this tab to select calculation options to use.

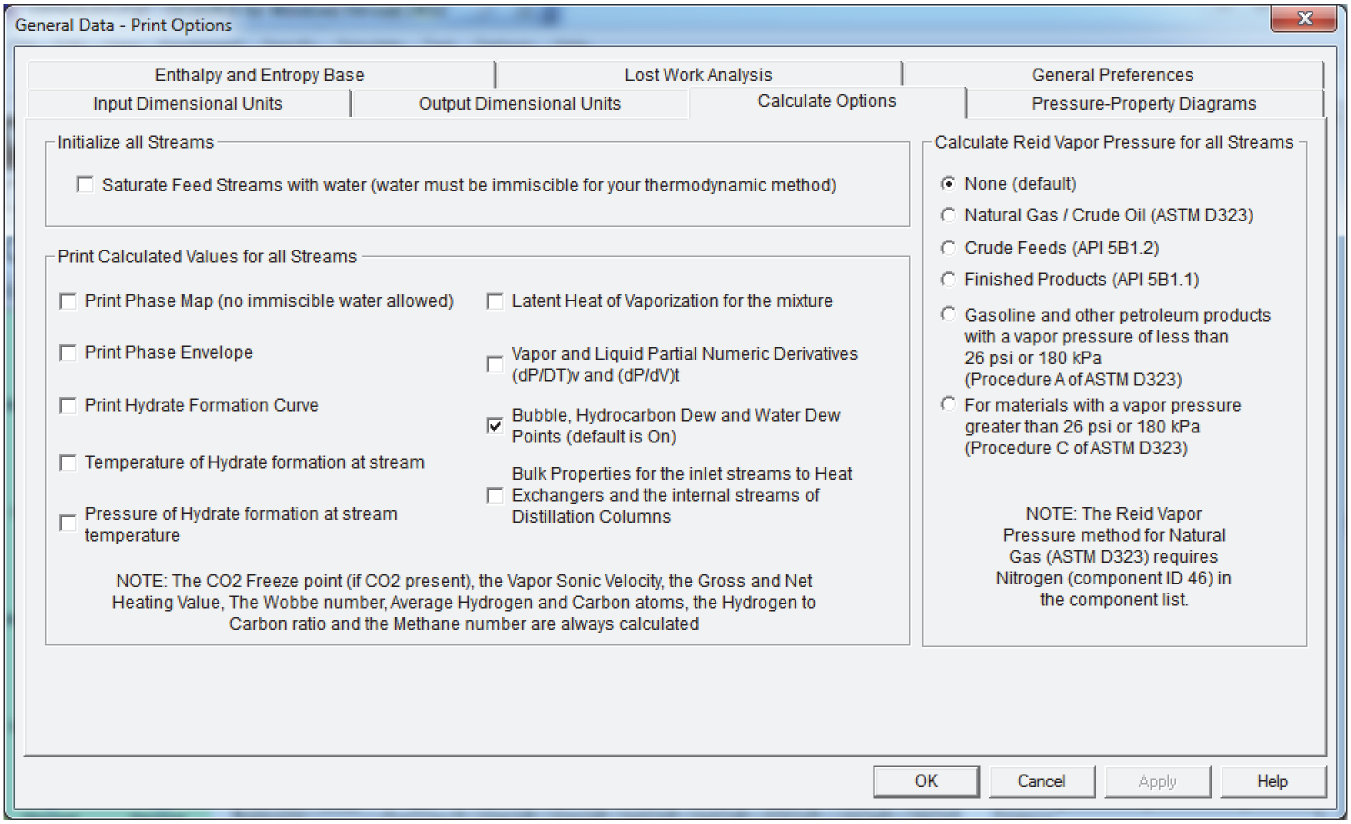


Figure 4: Calculate Options Dialog (from expander.psd)

Initialize All Streams

If you selected water as immiscible for your thermo method; then you can select to saturate feed streams with water (water must be immiscible for your thermodynamic method).

Print Calculated Values for All Streams

Select the desired options to print based on calculated values for streams:

- Print Phase Map (no immiscible water allowed)
- Print Phase Envelope
- Print Hydrate Formation Curve
- Temperature of Hydrate formation at stream pressure
- Pressure of Hydrate formation at stream temperature
- Latent Heat of Vaporization for the mixture
- Vapor and Liquid partial Numeric Derivatives (dP/DT)v and (dP/dV)t
- Bubble and Dew Points (default is On)
- Bulk Properties for the inlet streams to Heat Exchangers and the internal streams of Distillation Columns

The CO2 Freeze point (if CO2 is present), the Vapor Sonic Velocity, the Gross and Net Heating Value, the Wobbe number, Average Hydrogen and Carbon atoms, the Hydrogen to Carbon ratio and the Methane number are always calculated.

Calculate Reid Vapor Pressure for All Streams

Select either:

None
Natural Gas/Crude Oil (ASTM D323): this option requires Nitrogen, component ID 46, to be included in the list of components.
Crude Feeds (API 5B1.2)
Finished Products (API 5B1.1)
Gasoline and other petroleum products with a vapor pressure of less than 26 psi or 180 kPa (Procedure A of ASTM D323)
For materials with a vapor pressure greater than 26 psi or 180 kPa (Procedure C of ASTM D323)

Pressure Enthalpy Diagram

Use this tab to select whether to print a pressure enthalpy diagram and optional temperatures/pressures to use.

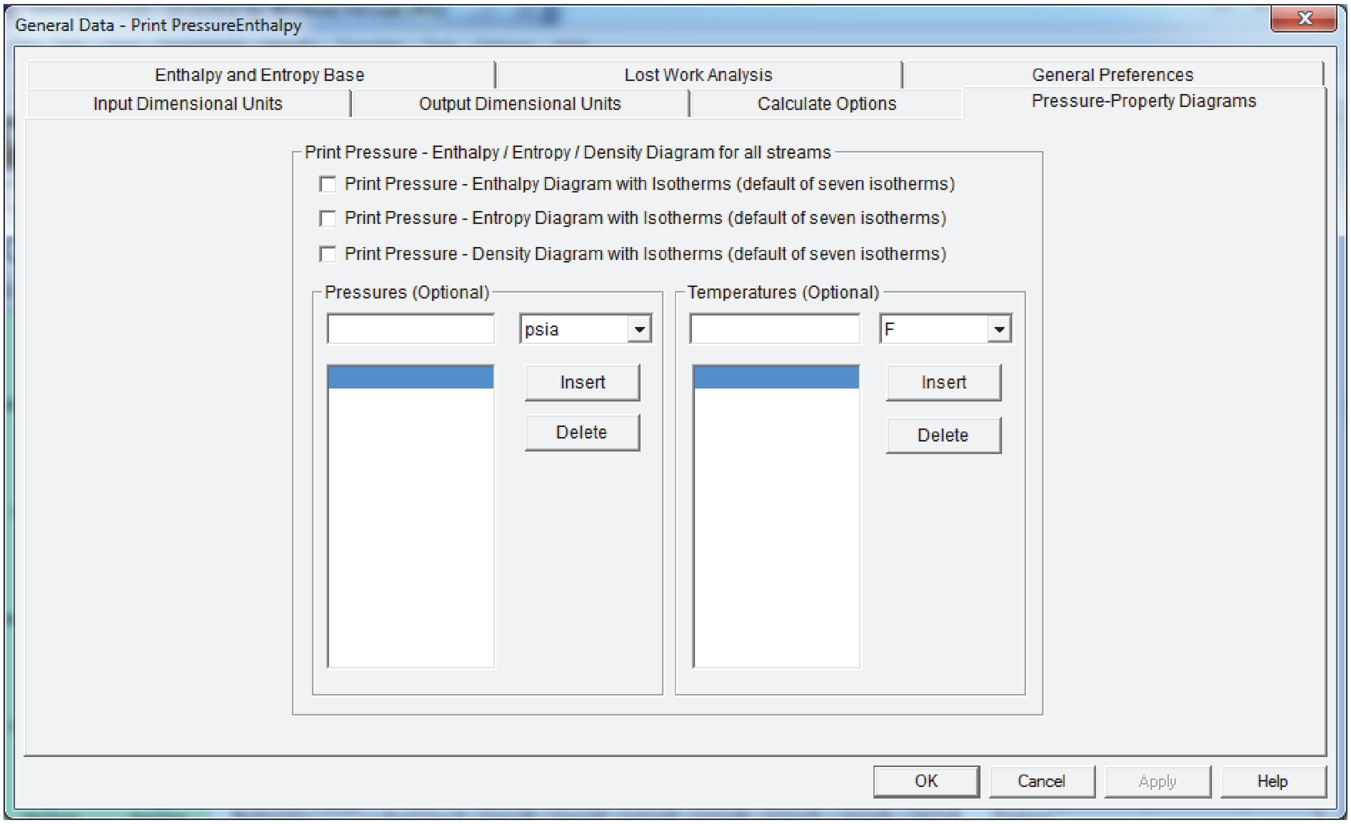


Figure 5: Pressure-Enthalpy Diagram Dialog (from expander.psd)

Print Pressure - Enthalpy Diagram for all streams

Print Pressure-Enthalpy Diagram with Seven Isotherms: Select this checkbox to include a pressure-enthalpy diagram for all streams in the flowsheet. The phase envelope diagram will be automatically generated first.

You can optionally enter the pressures and temperatures to use. If you do not put in pressures; then the pressures will be automatically generated starting at 14.7 psia, 50 psia, 100 psia, 150 psia and up to 120% of the top pressure for the bubble point or dew point. If you do not put in isotherm temperatures; then seven isotherm temperatures will be generated from the lowest bubble point temperature to 120% of the highest dew point temperature.

To enter pressure and temperatures values:
Click on the pressure on the list box below the Pressure column name; a blue highlight appears in the box. You can now type the desired value. Click the Insert button to add the value. Repeat this for the Temperature value.

You can also select the units to use for Pressure and Temperature.

Enthalpy and Entropy Base

Use this tab to enter specifications for the enthalpy and entropy base.

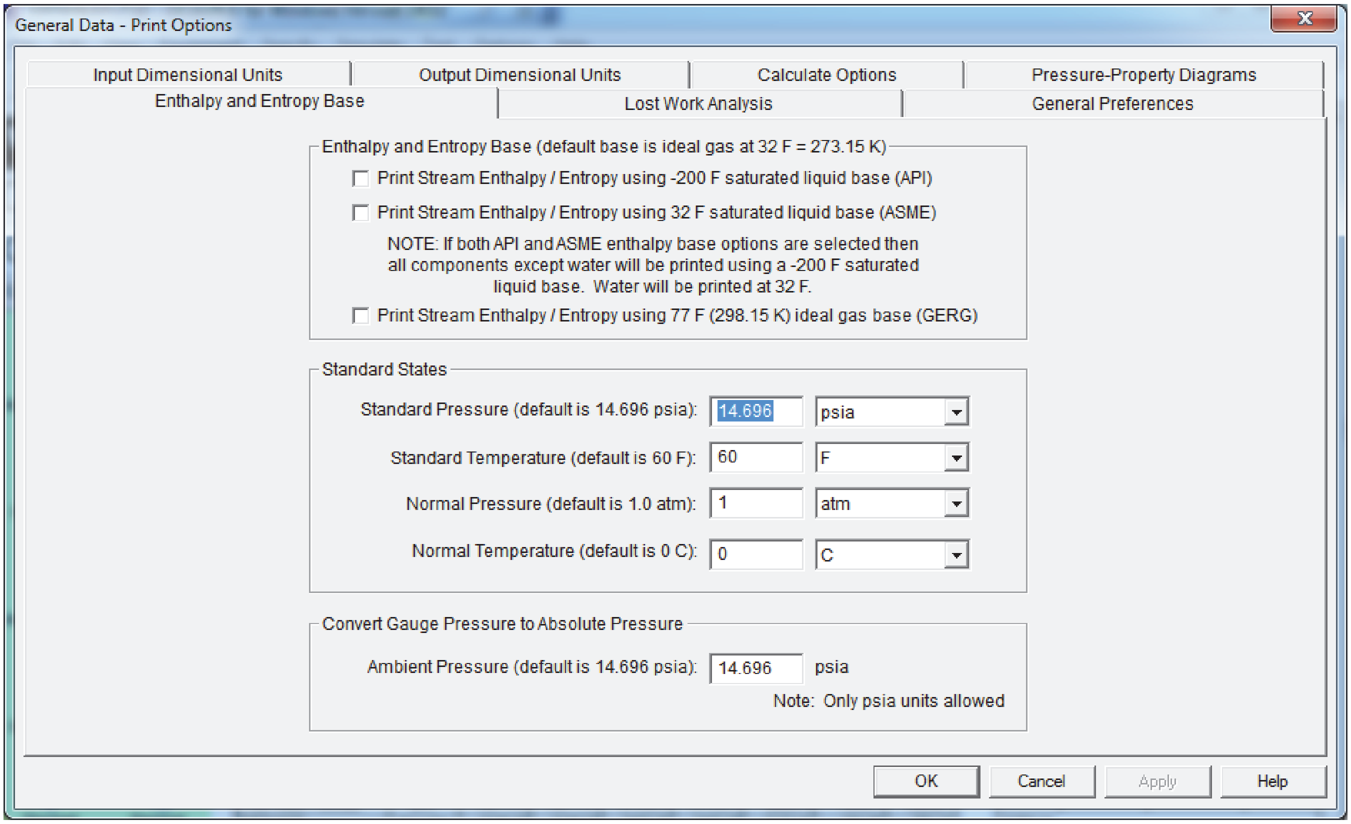


Figure 6: Enthalpy and Entropy Base Dialog (from expander.psd)

Enthalpy and Entropy Base (default is ideal gas at 32 F)

You can select to print:

Stream Enthalpy/Entropy using -200 F saturated liquid base (API)

Stream Enthalpy/Entropy using 32 F saturated base (ASME).

If you select both options; then all components except water are printed using -200 F saturated liquid base and water is printed at 32 F.

Stream Enthalpy/Entropy using 77 F (298.15 K) ideal gas base (GERG)

Standard States

You can define the standard conditions for reporting product volumetric flow rates. Enter the following states:

Standard Pressure/Temperature (default is 14.696 psia/60 F): enter the pressure/temperature for volume calculations at standard conditions.

Normal Pressure/Normal Temperature (default is 1.0 atm/default is 0 C): enter the pressure/temperature for volume calculations at standard conditions for metric/SI units

Convert Gauge Pressure to Absolute Pressure

Enter the ambient pressure. Units are PSIA. If the ambient pressure is not 14.696 PSIA (sea level/default), enter the desired value. Any pressure entered as PSIG is converted to PSIA by adding the ambient pressure as follows:

PSIG + ambient pressure = PSIA

Lost Work Analysis

Use this tab to select to perform a lost work analysis and enter specifications to use.

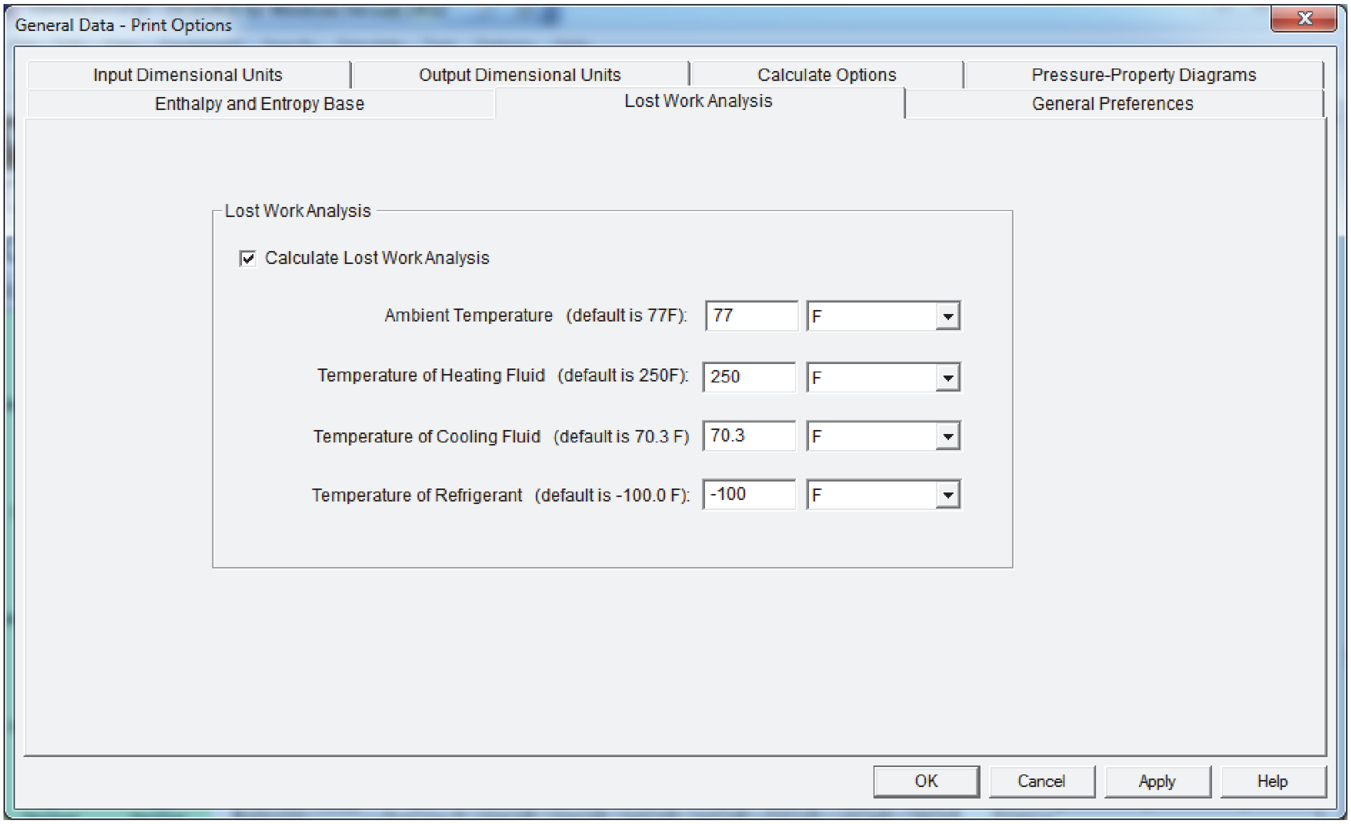


Figure 7: Lost Work Analysis Dialog (from expander.psd)

Calculate Lost Work Analysis

Select this checkbox to perform the calculation .Then you can specify:

Ambient Temperature (default is 77 F)
Temperature of Heating Fluid (default is 250 F)
Temperature of Cooling Fluid (default is 70.3 F)
Temperature of Refrigerant (default is -100.0 F)

Setting Options

To set various drawing options:

1. Open the **Options** menu.
2. Drag the highlight over the desired option ; then release the mouse button.

NOTE: A checkmark next to a menu item means the item is selected.

Ellipses (. . .) after a menu item signify a dialog box will display after you select the item.

You can select the following options.

Ortho Lines

Turns orthogonal line constraints on or off. When this is turned on, you can only draw lines running vertically or horizontally. If it is turned off, you can draw skewed lines in any direction.

Line Clash Detection

Turns on or off an option when any lines on the drawing cross each other, a short gap is left in the vertical line to show the crossing lines are not connected. This is useful for producing clear and easy to read drawings.

Show Snap Points

Displays or hides equipment snap points. Turning this option on causes equipment snap points to appear on the equipment symbols.

Secondary Snap

Turns on or off an option that automatically connects lines with symbols; it is designed to work when the Show Snap Points command is turned off, but you want to snap lines to the edges of symbols. When the Secondary Snap command is turned on, DESIGN II for Windows will try to snap any lines you draw to the sides of the box for the equipment symbol, rather than to the snap points.

Snap Symbols to Lines

Turns on or off an option that snaps both equipment symbols and arrows on to lines if you drop them close enough to a vertical or horizontal line.

NOTE: Neither symbols nor arrows can be snapped to non-orthogonal (skewed) lines.

Color

Turn color on or off for all items in the flowsheet drawing area (equipment symbols, streams, etc.)

Set Default Colors

Displays a dialog which allows you to select a flowsheet drawing item from a drop down list; then select a new color to use for this item. Click OK when done.

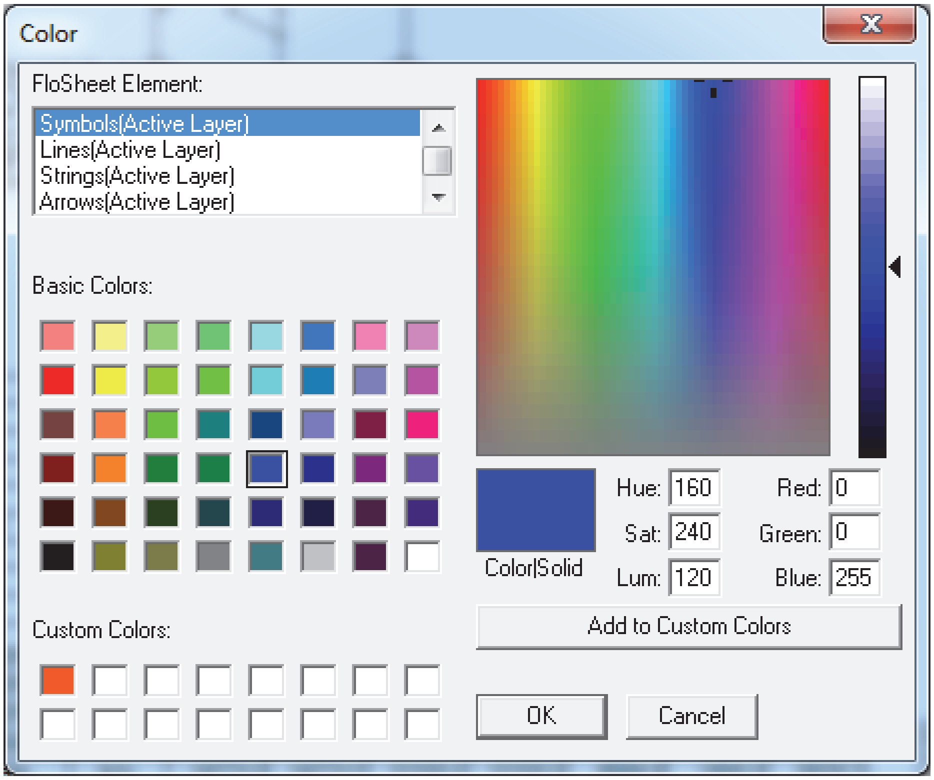


Figure 8: Color Dialog (from expander.psd)

Outlines

Turns on or off the option which displays equipment symbols, and groups of symbols and lines, as a bitmap when they are moved, or as a simple rectangle (or outline). Drawing is faster when the Outlines option is on.

Draw-Overwrite

This turns on or off an option, which uses Windows tricks to speed up drawing by orders of magnitude. When not selected, drawing is faster but can look untidy.

For example, arrows drawn on top of lines may appear with white space where the line passes underneath the arrow. DESIGN II for Windows automatically cleans up the drawing before it is printed. Turn off this option to speed up the drawing process when dealing with complex drawings.

Arrow Default Options

This allows you to select how arrows are automatically placed when you draw streams.

How Many Arrows:

Select how many arrows to place on a stream, either on the first segment only, on middle segments only, on last segment only, or on all segments (a stream can be drawn in one or more segments; if you are drawing a stream and click once; then move the stream in a new direction ; then click again, that is one segment).

Place Arrows where:

Select where to place an arrow on a stream, at the beginning, middle, or end of the line segment (determined where you start and stop drawing the stream).

Arrow Size:

Select to use regular sized arrows (1X), or larger sized arrows (2X, 3X, or 4X).

Auto-Routing

Turns Auto-Routing on or off. When Auto-Routing is on, moving an equipment symbol (or symbols) causes the attached streams to move with the equipment symbol(s). When off, any equipment symbol you move is moved by itself; connected streams do not move with it.

View Topology Errors

Currently not enabled.

Arrow Snap Warnings

This turns on or off a warning message display when arrows are not snapped to lines.

Show File New Dialog at Startup

Turns the display of the File New dialog on or off at startup. Default is On. The value is stored in the c:\designii\designii.txt file for reference every time the program starts.

Activate Undo / Redo

Turns the display of the File New dialog on or off at startup. Default is On. The value is stored in the c:\designii\designii.txt file for reference every time the program starts.

Activate Tooltips

Turns the Undo / Redo buffer on or off at startup. Default is On. The value is stored in the c:\designii\designii.txt file for reference every time the program starts.

Activate Commas for Numbers

Turns the display of commas for every 3 numbers in all displayed value on the flowsheet on or off at startup. Default is On. The value is stored in the c:\designii\designii.txt file for reference every time the program starts.

Activate Multiple Results Windows

This toggles on or off the display of results windows for streams and equipment after a simulation run. If it is on, each stream and equipment will have its results displayed in its own window. If it is off, there is only one results window with all stream/equipment results.

Activate Multiple Click Stream Creation

This toggles on or off whether streams are drawn use the old process (if you turn this option on, you must click to start a stream then click to end it) or the new process (hold down the mouse button to start a stream and drag the cursor to draw the stream).

Import DESIGN II Input File

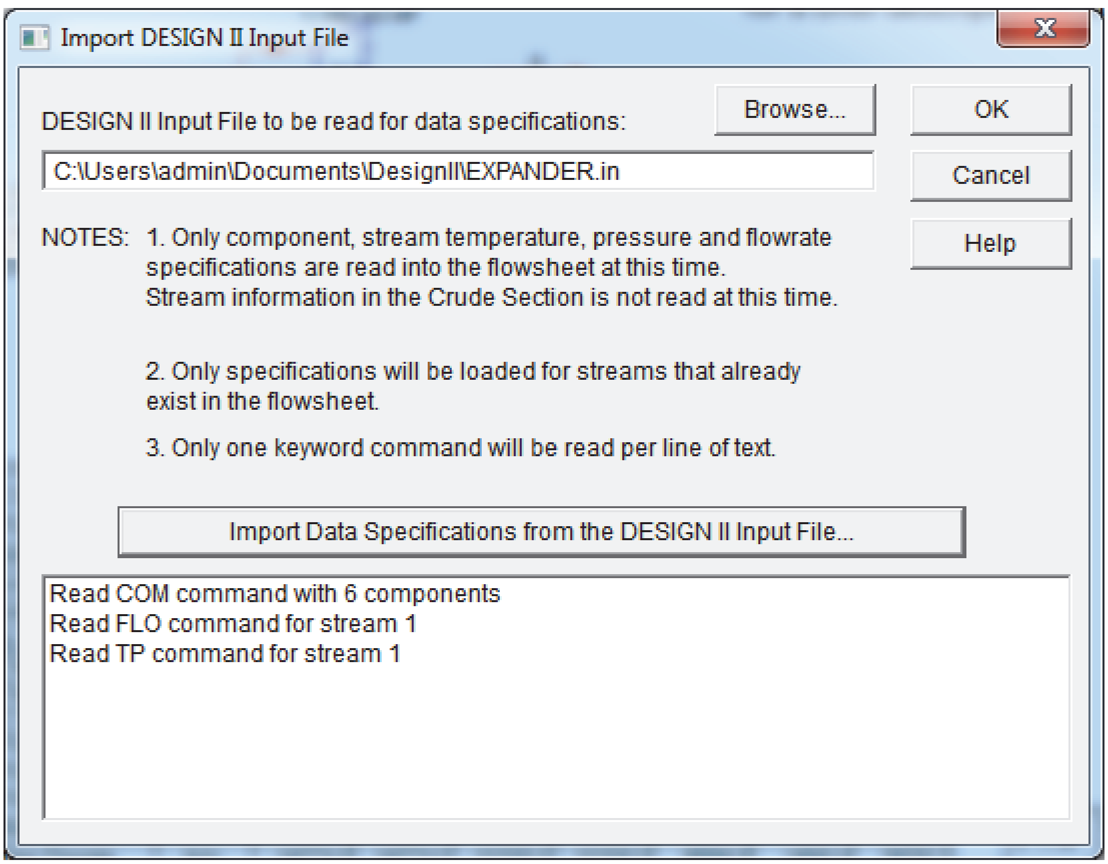


Figure 9: Import DESIGN II Input File (from expander.psd)

This dialog is used for reading an existing DESIGN II input file. Selecting Import Input File on the File pull down menu accesses the dialog. The keyword commands read at this time are component specifications (COM, PC, TC, VC, NAM, API, AMB, AMW) and stream specifications (FLO, TOT FLO, TP). A stream must exist for specifications to be read in for it.

Chapter 5: Equipment Modules

Working With Equipment

DESIGN II for Windows represents process equipment using standard engineering symbols. Currently, DESIGN II for Windows supports 36 equipment types with two generic symbols for creating your own equipment types. For a complete list of these equipment types, see the ***Choosing an Equipment Type*** section later in this chapter.

You construct your flowsheet by adding process equipment symbols to the flowsheet. You can use the Browser, tool bar, or Equipment menu to select the desired equipment (see the ***Adding Equipment*** section later in this chapter). You can then turn on a grid and/or rulers to help place equipment on the flowsheet.

When you place a piece of equipment on the flowsheet, the program automatically assigns a name and a number to it. The equipment symbols are numbered in the sequence you place them. However, you can enter a new name or number using the Equipment dialog. This dialog is displayed whenever you choose to set specifications for an equipment symbol (see ***the Setting Equipment Specifications*** section later in this chapter for details).

After placing an equipment symbol on your flowsheet, you enter specifications for it. These specifications become linked to the equipment symbol. If you copy or move the symbol, the specifications will move with it.

Once you have placed your equipment symbols, you can connect them with streams (see the ***Streams*** chapter for details). All equipment symbols have snap points along their edges, showing you where to connect streams. You can display snap points by selecting the **Show Snap Points** option under the **Options** menu.

You can also edit a piece of equipment, by either moving, copying, resizing, or deleting it (see the ***Editing Equipment*** section later in this chapter for details); or changing its specifications (see the ***Setting Equipment Specifications*** section later in this chapter for details).

You can also select two or more equipment symbols at a time to change the display results and thermodynamic options. You can also make limited changes to two or more equipment symbols at a time. Select the equipment symbols on the flowsheet (either hold down the Ctrl key then click on each equipment symbol) ; then right click to view a menu; choose **Common Equipment Properties** from the menu.

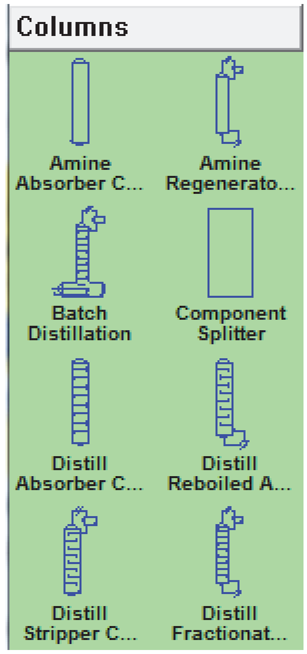
The **Common Equipment Properties** dialog appears which you can use to set the display results for the selected equipment symbols (Display Results on Flowsheet Equipment Name and Equipment Number). You can also choose the Thermodynamics tab to select common thermodynamic options for the selected equipment symbols. If one of the symbols you selected was a two-stream exchanger, a third tab appears on the dialog for setting the Thermodynamics on the tube side.

Details

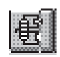
Please see the online DESIGN II Help, topic ***Equipment*** or the DESIGN II Unit Module Reference Guide ***Chapter 1: Equipment*** for more details on any equipment.

Adding Equipment

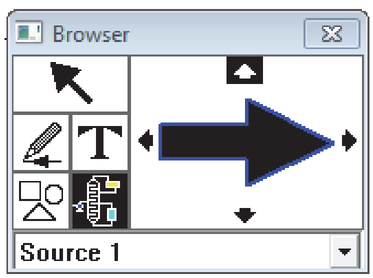
To add equipment to your flowsheet, you must first click on the desired equipment type on the Toolbox, or open the Equipment menu. Options: click on the **Equipment** tool of the **Browser** (if you are using it) or the **Equipment** button on the toolbar.



Toolbox equipment types



Equipment button



Equipment tool on the Browser

This places DESIGN II for Windows in the Equipment mode; all functions you perform in this mode will relate to working with equipment until you switch to another mode (such as Stream or Text mode). The cursor changes to a crosshair when positioned over the drawing area.

Before adding equipment, you should:

Turn on the Grid for alignment (open the **Options** menu and select **Grid**).

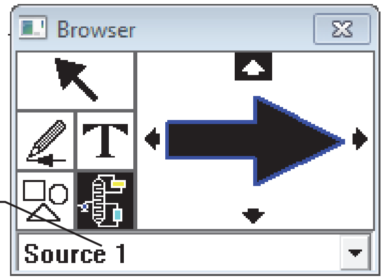
Turn on Snap Points for alignment and stream placement (open the **Options** menu and select **Show Snap Points**).

To add equipment to the flowsheet:

1. Click on the desired equipment type on the Toolbox, or open the **Equipment** menu from the toolbar; choose the desired type of equipment.
2. Choose the desired equipment type (see the ***Choosing an Equipment Type*** section later in this chapter for a full list of types).

   Note: If you selected the Equipment tool from the Browser, open the drop-down equipment list at the bottom of the Browser.

Equipment drop down list on the Browser



Scroll through the list until you find the desired equipment type; then click on it (an option is to drag the highlight down the list until it is over the desired type; then release the mouse button). The symbol for the selected equipment type displays on the Browser. OPTION: To change the orientation of the symbol, move the cursor over the equipment symbol displayed on the Browser and click on one of the four direction arrows. The symbol's orientation changes to reflect your selection. Any symbols you place after this will use the indicated orientation until you change the orientation again.

3. Move the cursor to the location on the drawing where you want to place the equipment ; then press the left mouse button. The equipment symbol will appear at this location, centered on the crosshair cursor.

4. OPTION: To move the symbol you just placed, place the crosshair over the symbol and click the left mouse button. You can now either:

    - edit the equipment symbol you just placed,
    - set the specifications for the equipment symbol
    - add another equipment symbol of the same type (repeating step 3),
    - change to another equipment type (following steps 1-3 again), or
    - change to another mode (such as Stream, Text, etc.)

Choosing an Equipment Type

Equipment on the Equipment menu or the browser equipment list is grouped by type. If you add any custom symbols (using the **Create Custom Symbol from Selected Graphics** feature), they will appear in the list if the custom symbol library is loaded using the **File/Manage Custom Symbol Libraries**. You can select the following equipment types (listed alphabetically):

Equipment Type	Purpose
Add Module	Models a user defined equipment using Inline Fortran
Air Cooler	Models a simple heat exchanger using air as the utility fluid
Air Cooler (2 Strm)	Models a simple heat exchanger using specified airflow stream
Amine Column Absorber	Models Amine Absorber Column with 1 or 2 feeds and 2 outlets
Amine Column Regenerator	Models Amine Regenerator Column with 1 feed and 2 or 3 outlets
Batch Distillation	Models a batch distillation column process
Compressor, Compressor 2	Models centrifugal and reciprocating compressors
Component Splitter	Non-equilibrium separation of individual components or chemicals
Controller	Adjust equipment specifications to meet a setpoint or pass equipment information to another unit module
CSTR	Continuous stirred tank rate-based, single-phase reactor model
Depressurizer	Model vessel/valve blowdown
Distill 1	Absorber column; no reboiler or condenser
Distill 2	Absorber column with reboiler
Distill 3	Stripper column with condenser
Distill 4	Fractionator column with reboiler and condenser
Divider	Splits a stream into two streams of the same composition; flowrate or fraction specs

Divider (Multiple)	Splits a stream into two or more streams of the same composition; using multiple flowrate or fraction specs
Double Pipe HX 1, 2	Uses double pipe heat exchanger symbols (one shell and two shell); the heat exchanger module calculations are used
Equilibrium Reactor	Models gas-phase reactions
Exchanger 1	Heat exchanger with one process stream
Exchanger 2, 3, 4	Heat exchanger with two counter-current process streams
Expander	Gas or liquid expanders or turbines
Expander Compressor	Passes the calculated work by the expander to the compressor as the maximum available work
Expander Pump	Passes the calculated work by the expander to the pump as the maximum available work
Fired Heater	Models heater duty and fuel consumption
Flash 1	Vertical vessel phase separation; set stream conditions
Flash 2	Horizontal phase separation; set stream conditions, with boot
Flash 3	Horizontal phase separation; set stream conditions, without boot
Flash 4	Vertical vessel phase separation without a mist eliminator
Flow Meter	Models an orifice plate based on flow rate, size or delta pressure; single or two-phase flow allowed
Generic 1	A large symbol for user added modules and equipment modules topology not currently supported graphically (obsolete; kept for older version compatibility)
Generic 2	A small symbol for user added modules and equipment module topology not currently supported graphically (obsolete; kept for older version compatibility)
Hydrotreater	Models single or multibed hydrotreating reactor; includes quench stream calculation
Line	Models hydraulic pressure drop calculations in process lines; allows heat transfer to surroundings; multiphase calculations
LNG Exchanger	Models multistream heat exchange; Maximum of 3 shellside streams
LNG 11x11 Exchanger	Models multistream heat exchange; Maximum of 11 shellside or tubeside streams (up to 20 total)
LNG 19x11 Exchanger	Models multistream heat exchange; Maximum of 19 shellside by 11 tubeside streams (up to 20 total)
Mass Balance	Control recycle flowrate and / or calculate recycle makeup flowrate
Mixer, 2, 3	Models adiabatic combination of streams
MultiFlash	Models flashing one composition to several outlet conditions or several streams to set outlet conditions
MultiPhase Flash	Models rigorous three-phase stream separation
Packed Absorber	Provides preliminary calculations for a packed column by using a number of sections, HETP per section, height per section and pressure drop per unit height per section
PFX 1 (3x3)	Models Plate Fin Heat Exchanger with up to 3 hotside streams and 5 coldside streams
PFX 2 (5x5)	Models Plate Fin Heat Exchanger with up to 5 hotside streams and 5 coldside streams
PFX 3 (9x9)	Models Plate Fin Heat Exchanger with up to 9 hotside streams and 9 coldside streams
Phase Envelope	Models calculation of bubble and dew point lines at specified pressures; includes mixture critical point
Phase Map	Models calculation of bubble and dew point lines; allows up to 5 liquid fraction lines; includes mixture critical point, cricondentherm and cricondenbar

Plug Flow Reactor	Rate based model of single-phase plug flow reactions
Polytropic Compressor	Models a centrifugal compressor or pump using manufacturer's curves
Pump	Model for centrifugal and reciprocating pumps
Reactor	Models a stoichiometric reactor or set reactions, such as CO Shift, methanation, methanol synthesis, ammonia synthesis, and primary and secondary reformers
Refine 1, Refine 1A	Models crude distillation column with partial or total condenser
Refine 2, Refine 2A	Models crude distillation column with no external condenser
Refine 3, Refine 3A	Models crude distillation column with top tray pumparound and external condenser (no external reflux)
Ref: Pumparound	Pumparound addition for a Refine Column
Ref: Stripper (Reb)	Stripper with Reboiler Heat addition for a Refine Column
Ref: Stripper (Stm)	Stripper with Steam Heat addition for a Refine Column
Shortcut Fractionator	Models a partial or total fractionator column using key component recoveries; estimates reflux ratio, feed tray location and number of stages
Sink 1	Shows a sink symbol for a stream to anchor to, no practical use
Source	Shows a source symbol for a stream to anchor to, no practical use
Spreadsheet	Description
Stream Manipulator	Modifies or copies a stream's composition
Tank	Calculates the dynamic behavior of feed, accumulations, and products from a constant volume tank.
Valve, 1, 2, 3, 4	Models adiabatic pressure reduction of a stream

Equipment list (by type)

Columns

Amine Absorber Column

Columns with batch or continuous feeds for batch processes

Amine Regenerator Column

Columns with a partial condenser and a reboiler

Batch Distillation

Columns with batch or continuous feeds for batch processes

Component Splitter

Splitter for separation of individual components or chemicals

Distillation Absorber Column, Distillation Reboiled Absorber Column, Distillation Stripper Column, Distillation Fractionator Column

Columns with multiple feeds/products, condensers, reboilers, etc.

Packed Absorber

Provides preliminary calculations for a packed column by using a number of sections, HETP per section, height per section and pressure drop per unit height per section.

Refine 1, 1A, 2, 2A, 3, 3A

Refine distillation columns

Refine Pumparound/Refine Stripper (Reb)/Refine Stripper (Stm)

Refine symbols you can add to your flowsheet

Shortcut Fractionator

Quick solving distillation column that uses default settings

Stream Manipulator

Manipulator for duplicating streams

Heat Exchangers

Air Cooler

Single-Stream Heat Exchangers using forced air for cooling or condensing of the process stream with a calculated air side stream

Air Cooler (2 stream)

Two-Stream Heat Exchangers using forced air for cooling or condensing of the process stream with a specified air side stream

Double Pipe HX 1, 2

Module with two double pipe heat exchanger symbols (one shell and two shell); the heat exchanger module calculations are used

Exchanger 1

Single-Stream heat exchanger

Exchanger 2, 3, 4

Two-Stream shell and tube heat exchanger

Fired Heater

Fired heater calculating fuel consumption and actual duty

LNG Exchanger, LNG 11x11 Exchanger, LNG 19x11 Exchanger

Heat exchanger allowing multiple shell and tubeside streams

PFX 1 (3x3)

Brazed Aluminum Plate Fin Heat Exchanger with up to 3 hot streams and 3 cold streams

PFX 2 (5x5)

Brazed Aluminum Plate Fin Heat Exchanger with up to 5 hot streams and 5 cold streams

PFX 3 (9x9)

Brazed Aluminum Plate Fin Heat Exchanger with up to 9 hot streams and 9 cold streams

Pressure Change

Compressor/Compressor 2

Reciprocating or centrifugal gas compressor

Depressurizer

Vessel and valve which models blowdown

Expander

Isentropic expansion of vapor, liquid, or vapor/liquid inlet streams

Expander Compressor

A utility module that passes or uses the calculated work by the expander to the compressor as the maximum available work

Expander Pump

A utility module that passes or uses the calculated work by the expander to the pump as the maximum available work

Flash 1, 2, 3, 4

Separator drum or simple flash calculation

Line

Module for pressure drop in transmission lines or plant piping

MultiFlash

Flash one composition to multiple outlet conditions

MultiPhase Flash

Rigorous three-phase separation

Polytropic Compressor

Simulate centrifugal compressor based on manufacturer's performance curves.

Pump

Reciprocating or centrifugal liquid pump

Valve, 1, 2, 3, 4, 5

Pressure letdown valve

Reactors

CSTR

Continuous Stirred Tank Reactor allowing multiple simultaneous reactions

Equilibrium Reactor

Vapor phase reaction using Gibbs free energy minimization technique

Hydrotreater

Single or multiple bed hydrotreater model including quench streams

Plug Flow Reactor

Reaction calculation using kinetic information

Reactor

Stoichiometric reactor calculation

Stream Manipulator

Stream Operations

Component Splitter

Divider

Module for dividing an inlet stream into two outlet streams

Divider (Multiple)

Module for dividing an inlet stream into multiple outlet streams

Flash

Line

Mass Balance

Set the recycle flowrate or calculate recycle makeup flowrate

Mixer, 2, 3

Module for mixing multiple inlet streams

Multiple Flash

Multiple Phase Flash

Phase Envelope

Graph and table for phase envelope

Phase Map

Graph and table for phase map

Stream Manipulator

Valve 1, 2, 3, 4, 5

Controls and Metering

Controller

Varies flowsheet parameters until a setpoint is met or pass calculated information forward

Depressurizer

Flow Meter

Rigorous pressure drop calculations across an orifice

Tanks

Tank – Vertical

Calculates the dynamic behavior of feed, accumulations, and products from a constant volume tank.

Tank – Spherical

Calculates the dynamic behavior of feed, accumulations, and products from a constant volume tank.

Miscellaneous

Source 1

A visual representation (arrow) to indicate the source; this symbol currently has no other functionality

Sink 1

A visual representation (arrow) to indicate the sink; this symbol currently has no other functionality

Generic 1, 2

User-written FORTRAN models of equipment

Add Module

User-written FORTRAN models of equipment

Editing Equipment

You can move, copy, or delete symbols from your flowsheet.

NOTE: Before editing a symbol, make sure DESIGN II for Windows is in the Equipment mode.

Moving Equipment

1. Select the desired symbol by placing the crosshair anywhere within the symbol and clicking the left mouse button. The symbol changes color to indicate it has been selected.
2. Keep the crosshair inside the symbol; then press and hold the mouse button down. The crosshair changes to a hand.

3. Drag the symbol to its new location; then release the mouse button. If a stream is attached the equipment symbol and you have the **Auto-Routing** option turned on, the stream will move with the symbol. If **Auto-Routing** is off, the stream connection is broken and the symbol is moved by itself.
4. Once you've placed the symbol in its new location, you can deselect the symbol by placing the cursor in a blank area of the flowsheet and clicking the left mouse button.

Copying Equipment

You can copy an equipment symbol (or symbols) using the **Copy** function under the **Edit** Menu.

To copy an equipment symbol or symbols:

1. Click on the desired symbol(s), ; then release the mouse button.
2. Open the **Edit** menu and select **Copy**
3. Open the **Edit** menu and select **Paste**. A copy of the selected symbol is placed in the middle of the screen, while the original symbol remains in place.

Deleting Equipment

To delete a symbol:

1. Place the crosshair anywhere within the symbol and click the left mouse button. The symbol is selected.
2. Click the right mouse button. A message displays, asking if you want to delete the symbol. Click **Yes** to delete it, **No** to cancel the deletion.

   NOTE: Once you delete a symbol, it is permanently removed and you cannot recover it.

Resizing Equipment

You can scale a piece of equipment to one of four set sizes.

To resize an equipment symbol:

1. Place the crosshair anywhere within the symbol and click the left mouse button. The symbol is selected.
2. Click on the **Edit** Menu and select a **Change Symbol/Arrow Size**; then 1X, 2X, 3X, or 4X.

   NOTE: This size will remain in effect for the next symbol you place on the drawing. To deactivate the resizing, select the new symbol, click on the **Edit** menu and select the **1X** option.

Disabling/Enabling Equipment

You can disable (and enable) any piece of equipment in the flowsheet. You can also disable streams and sheets. All validation rules apply as if the disabled equipment does not exist. For instance, if a piece of equipment is disabled then all product streams from that equipment must now be specified as if they are feed streams to the process. Disabled equipment will have "Disabled" written across the equipment and an X placed across it.

1. To disable/enable equipment, right-click on the equipment.
2. Select Disable (or Enable) from the pop-up menu.

Setting Equipment Specifications

To set equipment specifications:

Place the crosshair anywhere within the symbol and double-click. Or, select the symbol and then choose the **Specify Item** under the **Edit** menu. The Equipment dialog box displays.

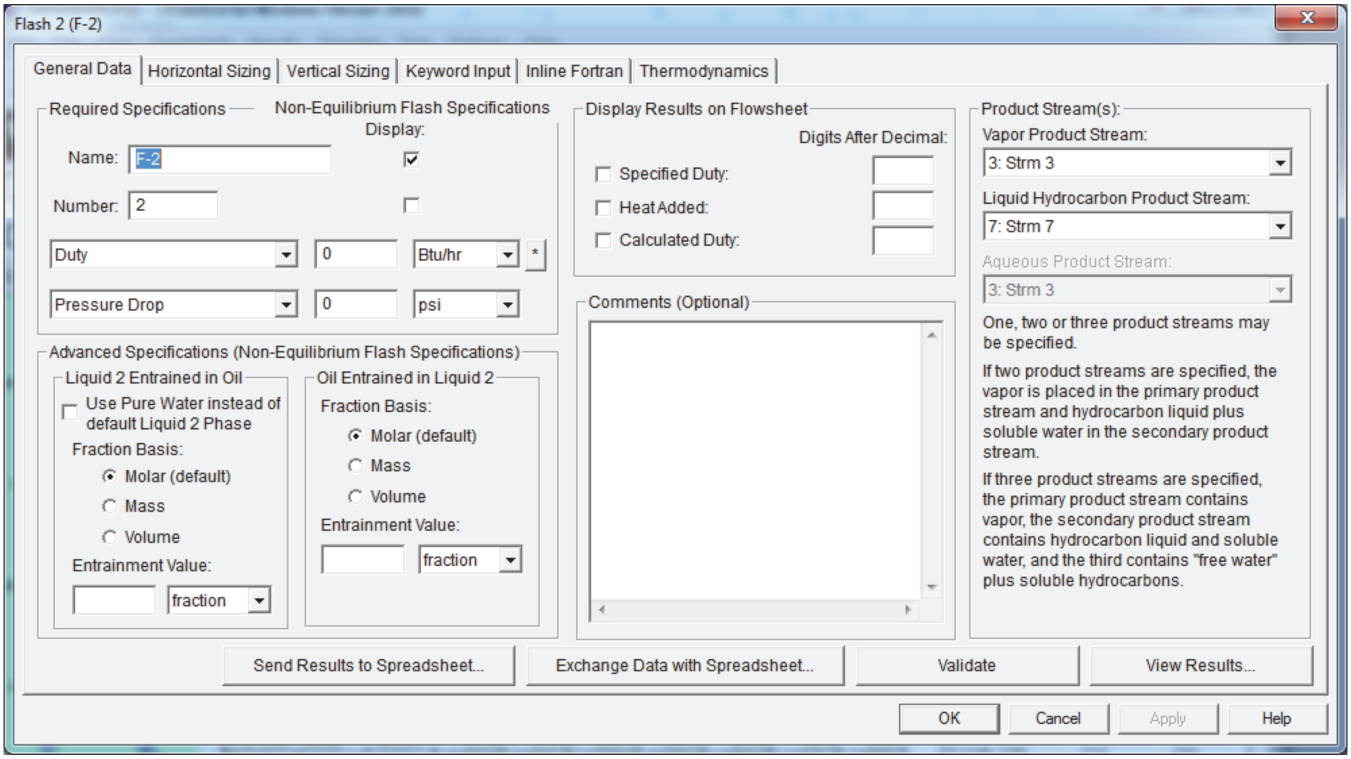


Figure 1: Example Equipment Dialog Box (from expander.psd)

1. Enter a new equipment name and/or number if desired. **Equipment numbers must be unique.** You can also click the **Display box** to show/hide the name and/or number next to the symbol on the flowsheet.

2. A dialog will have multiple tabs. All will have a **General Data** tab; the other types of tabs and the data you must enter depend on the type of equipment.

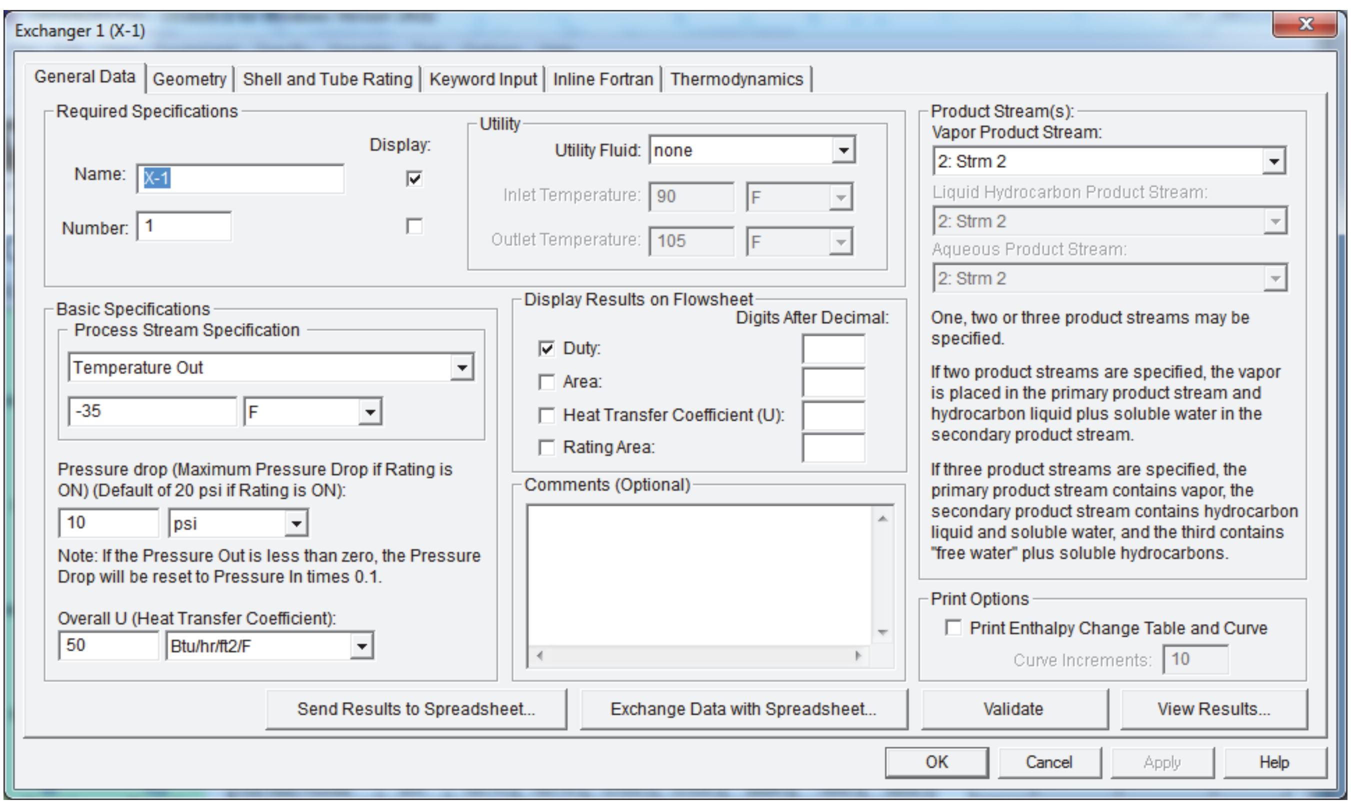


Figure 2: Example General Data tab from Exchanger dialog (from expander.psd)

Since the specifications for each equipment type vary, view the Help topic for the equipment type (click the **Help** button). An option is to set **Keyword Input** or **Inline Fortran** for the equipment; click the appropriate tab and type in text or click on the **Load Template** button. Many dialogs will have an optional comments field for you to enter notes about the equipment.

3. After completing the basic/optional specifications, click the **OK** button to close the dialog or Apply to save your changes and leave the dialog open.

Thermodynamics

The Thermodynamics tab (either on the Thermodynamic Methods dialog, equipment dialog, or stream dialog) provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see ***Chapter 7: Thermodynamics*** in the DESIGN II Reference Guide.

This dialog can be accessed via the Specify/ Basic Thermo menu option.

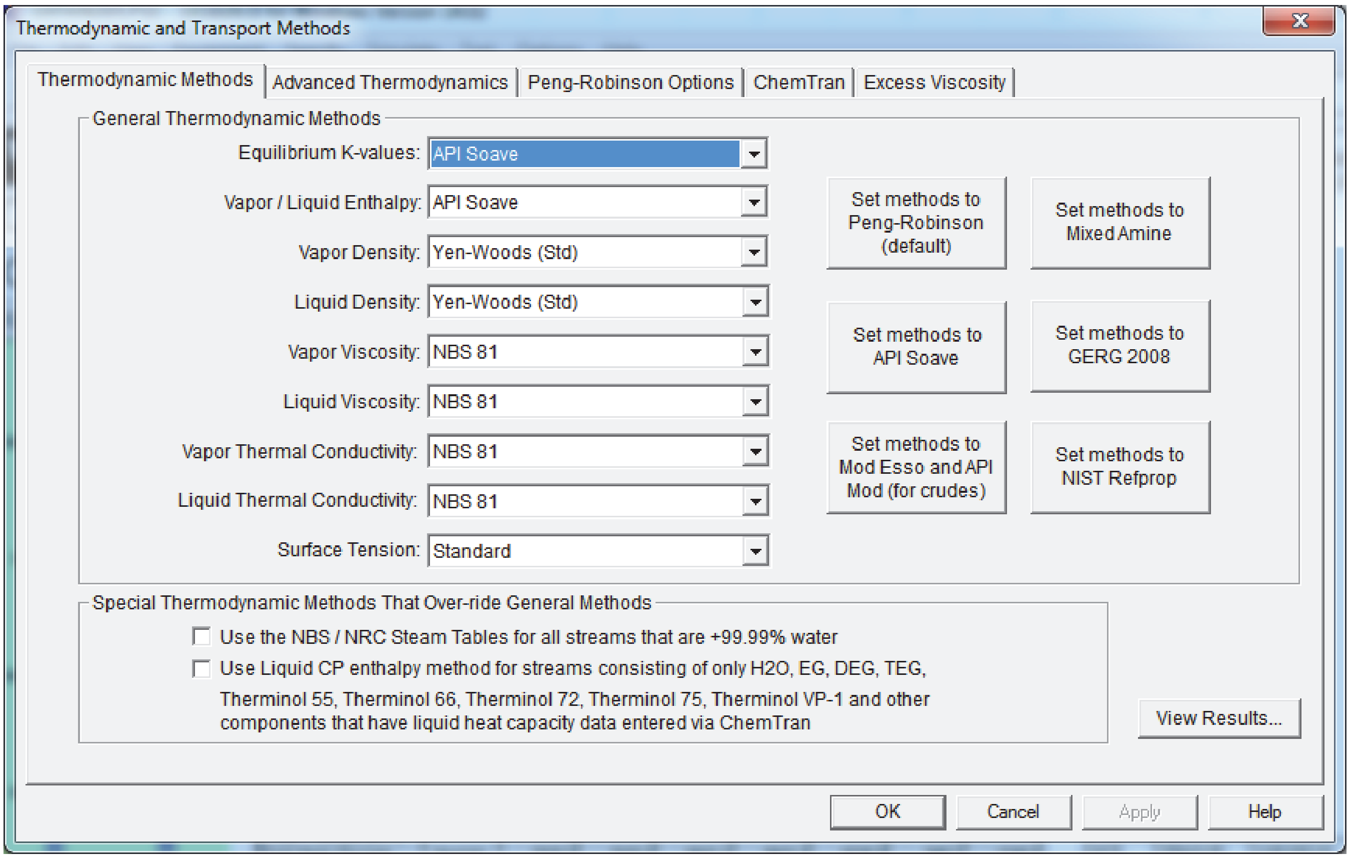


Figure 3: Thermodynamics tab on the Thermodynamics dialog (from expander.psd)

Keyword Input

DESIGN II for Windows provides dialogs for entering many of the key specifications for DESIGN II equipment models. However, it does not currently support all the possible specifications directly in the dialogs. Until all specifications are supported, templates of DESIGN II keyword commands are provided for equipment models.

You can access these templates by clicking the **Keyword** tab on an equipment dialog or the **Specify/Keyword Input** menu choice. When you select keyword dialog a window will appear that allows you to create keyword commands.

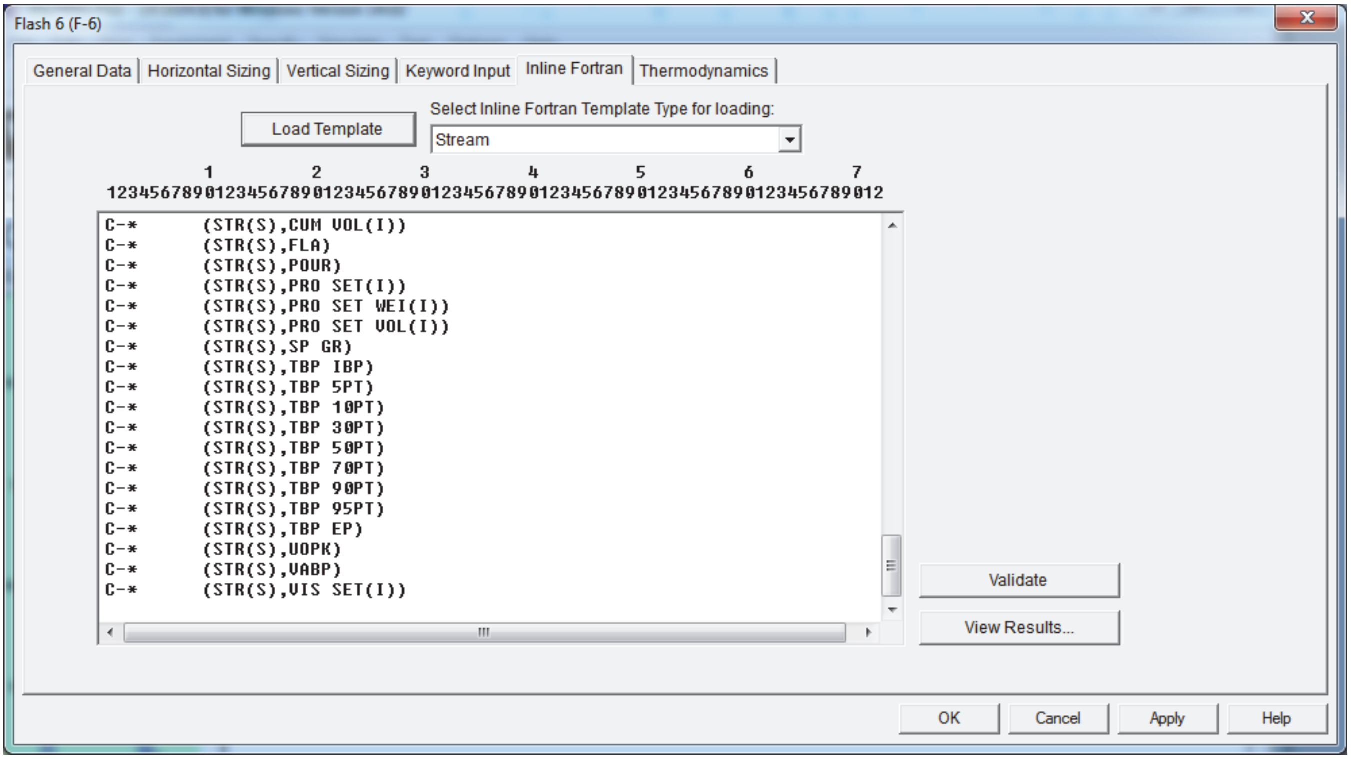


Figure 4: Keyword Input tab from the Flash dialog (from inlfrn1.psd)

Using Command Templates

Equipment command templates contain keyword commands for DESIGN II equipment models. When they first appear in the edit display, all lines of the template will begin with the characters 'C-*'. The '*' prevents these lines from appearing in your DESIGN II input. The 'C-' is the standard method for adding comment lines to your DESIGN II input.

The procedures for using these templates are:

Creating Keyword Commands	This is the procedure to use when you first create keyword commands from a command template.
Editing Keyword Commands	This is the procedure to use when you are modifying keyword commands that you have already created.

Creating Keyword Commands from Templates

When you first use a command-template dialog for an equipment, the template will contain all the commands for the equipment that are not supported by DESIGN II for Windows.

The procedure for creating commands for your input from the templates is:

1. Locate the keyword command that you need for your specification.
2. Remove the 'C-*' using keyboard editing keys (delete, backspace, page up, page down, arrow keys, etc.). You can use the mouse for highlighting text and working with the editing keys.
3. Edit the command template so that it contains your specifications and the correct dimensional units. Be sure to delete the characters that specify the type of dimensional unit (T units, P units, molar Q units/t units, etc.)
4. Remove the '*' from the 'C-' for any lines in the template that you want to appear in your DESIGN II input file as comments. Lines beginning with a 'C-*' will not appear in your DESIGN II input file.
5. You also may enter commands or comments directly into this file without using the template.
6. Exit the dialog when you have finished creating the keyword commands.

Editing Keyword Command Templates

In the command-template dialog, additional options are provided for editing a previously created set of keyword commands. When you enter the dialog, DESIGN II for Windows will display those commands with the command template. If you select the Load Templates button, a dialog with the three additional options described below is displayed. After execution of one of these options, use the procedure for creating keyword commands.

Yes Use this option if you want to replace the commands displayed with a new command template.

No Select this option to append a new command template for this equipment to the commands that are displayed. You are most likely to need this option if you delete lines from the command template.

Cancel This choice cancels this dialog and no action is taken.

Inline Fortran

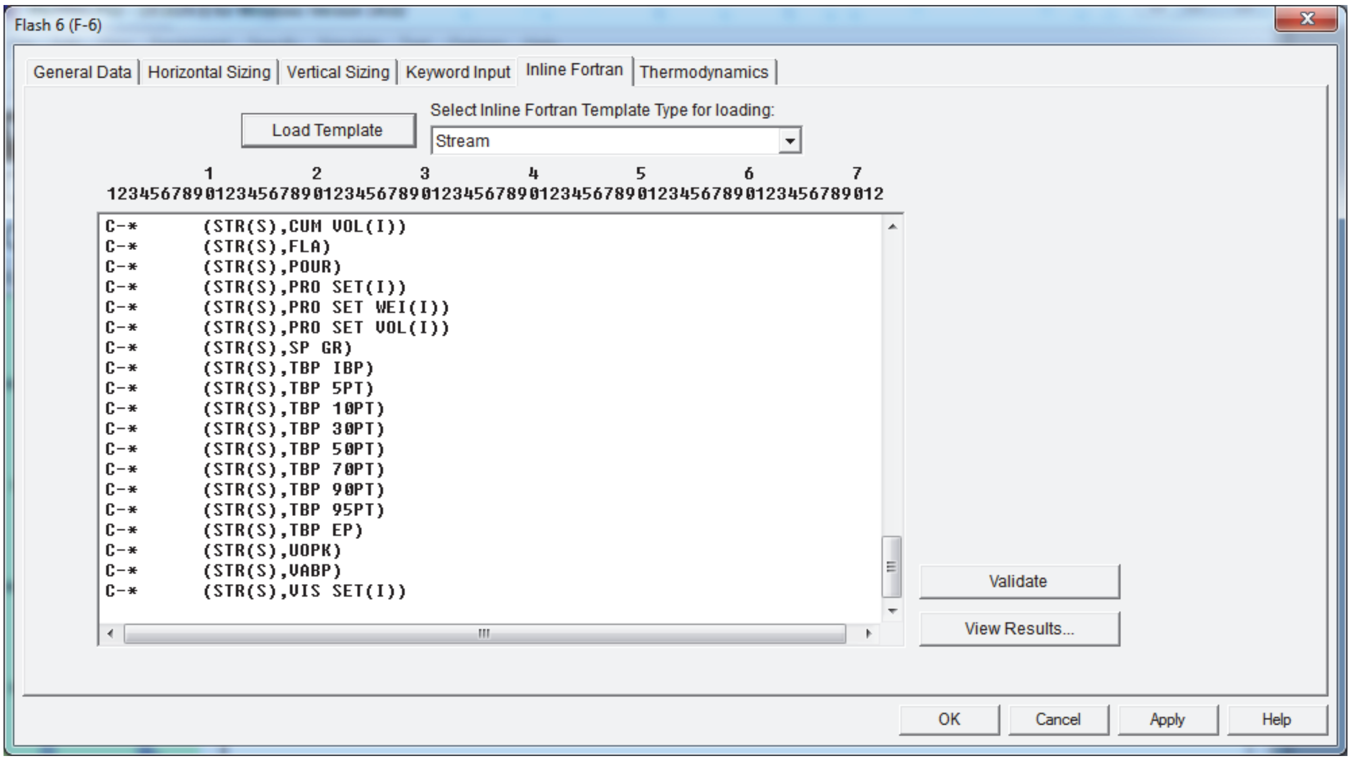


Figure 5: Inline Fortran Dialog (from inlfrn1.psd)

You can enter Inline Fortran statements using the Inline Fortran tab on equipment dialogs or the **Specify/Inline FORTRAN** menu choice. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Templates are available for the following equipment:

ADD Module
Component Splitter
Compressor
CSTR
Depressurizer
Distillation Column
Divider

Equilibrium Reactor
Expander
Fired Heater
Flash
Flowmeter
Heat Exchanger
Hydrotreater
Line Module

LNG Exchanger
Plug Flow Reactor
Polytropic Compressor
Pump
Refine Column
Shortcut Distillation
Stream Manipulator
Valve

Chapter 6.1: ADD Module

General

ADD blocks are user-written Fortran models of equipment that can be ADDed to DESIGN II. These blocks are usually highly specialized equipment models, but can also be used to perform general operations on stream and equipment variables. Inline Fortran now eliminates the need to perform the external compiling and linking steps to put an ADD block routine in DESIGN II. With this method the ADD block routines can be entered in the DESIGN II input file, along with the standard equipment modules. Compilation and linkage steps, which are now internal to DESIGN II, are automatic.

Details

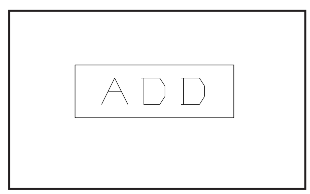
The ADD Module feature in DESIGN II allows you to use your own process technology in conjunction with the standard equipment modules. To use the ADD Modules(s), you must write your own Fortran subroutine(s) to interface with DESIGN II, which will perform the Additional calculations in the course of the simulation. Typical applications for ADD Modules include the following:

Economic Evaluations
Simulation of Proprietary Unit Operations
Substituting Unit Calculation Techniques
Pollution Calculations

Please see the online **DESIGN II** Help topic ***Equipment/ADD Module*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 2: ADD Module*** for more details.

Symbols

The ADD unit module has one symbol:



The Add Module requires that at least one inlet stream and at least one outlet stream be connected to the module.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

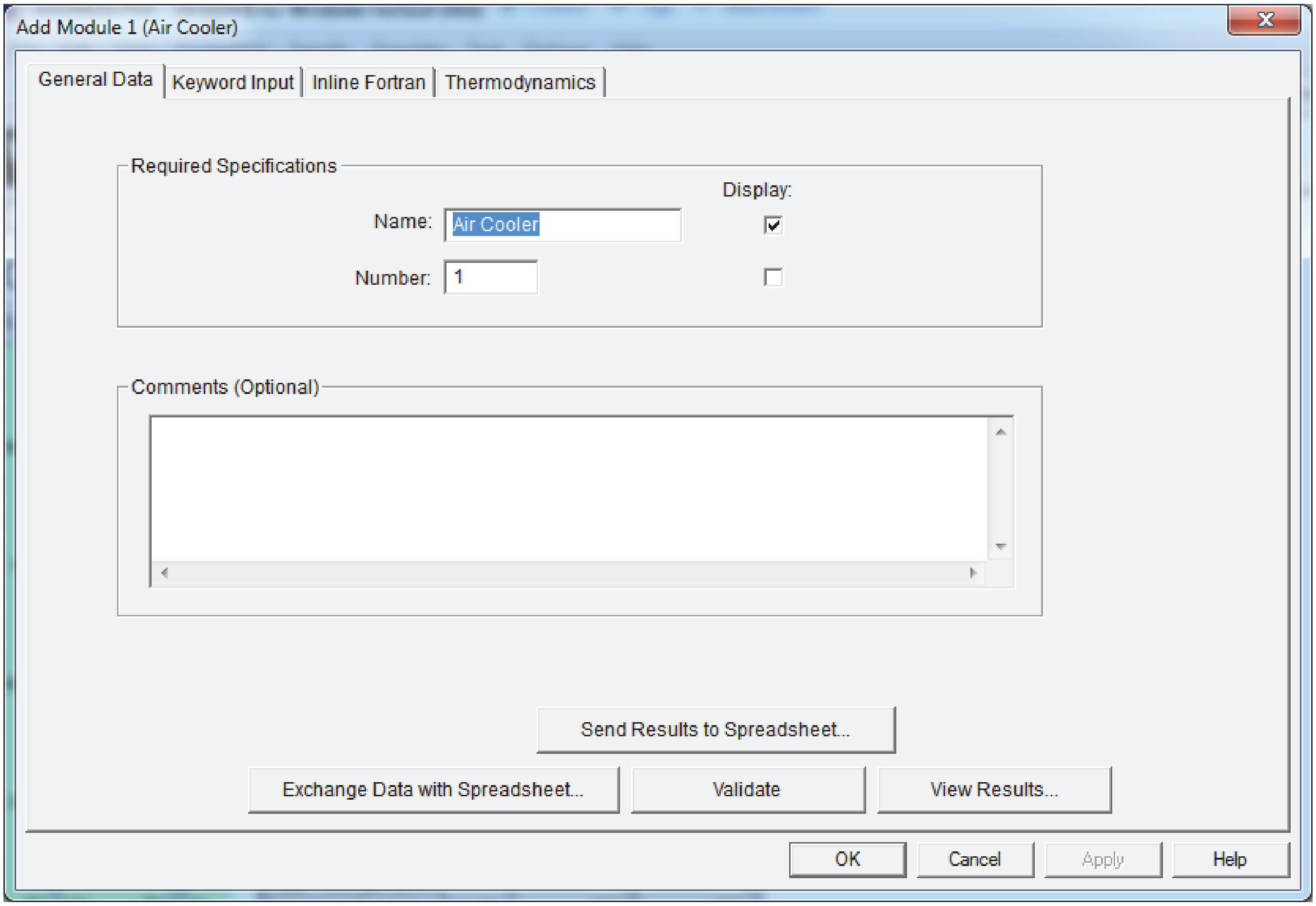


Figure 1: Add Module Dialog (from addmod.psd)

General Data

Use this tab to enter the basic specifications when creating a custom equipment module.

Name/display, Number/display	The name/number associated with the equipment. You can enter a name/number for the equipment; then choose to display it on the flowsheet.
Send Results to Spreadsheet, Exchange Data with Spreadsheet, Validate, View Results	Refer to Chapter 3: Basics for details.

Required Specifications: Keyword Input

DESIGN II for Windows provides dialogs for entering many of the key specifications for DESIGN II equipment models. However, it does not currently support all the possible specifications. Until all specifications are supported, templates of DESIGN II keyword commands are provided for equipment models.

You can access these templates by selecting the Keyword Input tab from the Add Module dialog. When you select keyword dialog a window will appear that allows you to create keyword commands.

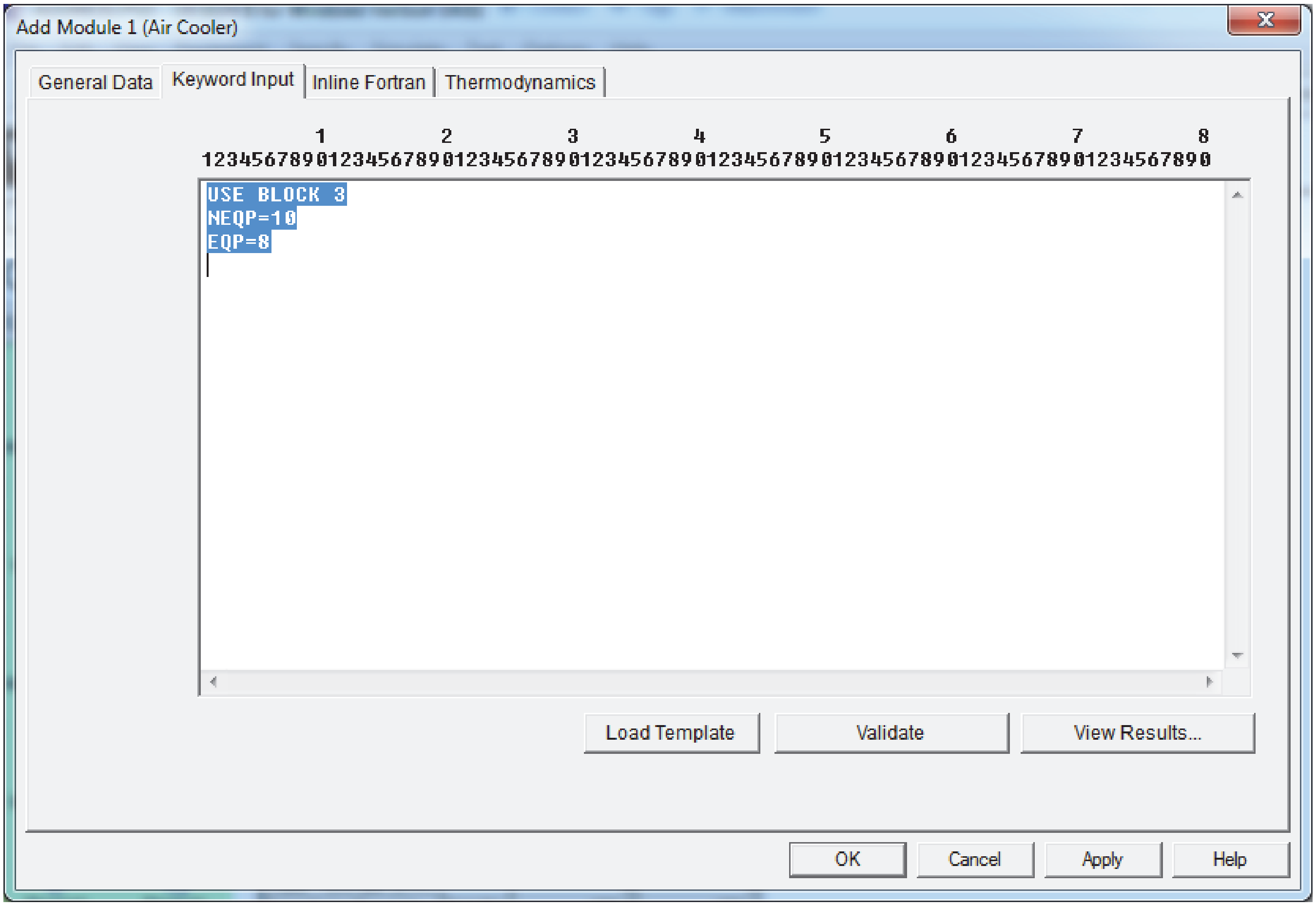


Figure 2: Keyword Input Dialog (from addmod.psd)

Optional Specifications: Inline Fortran

ADD blocks are user-written Fortran models of equipment that can be ADDed to DESIGN II. These blocks are usually highly specialized equipment models, but can also be used to perform general operations on stream and equipment variables. Compilation and linkage steps, which are internal to DESIGN II, are automatic. Use **the Inline Fortran** tab to enter this information.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Add Module Examples

There is a sample flowsheet in “Chapter 8: Add Module Samples - c:\designii\samples\equipmnt\add” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.2: Air Cooler

General

An Air-Cooled Exchanger (AIRC) is a device for rejecting heat from a fluid directly to ambient air. This is in contrast to rejecting heat to water ; then rejecting it to air, as with a shell-and-tube heat exchanger and a wet cooling tower system.

The obvious advantage of an Air-Cooled Exchanger is that it does not require water, which means that plants requiring large cooling capacities need not be located near a supply of cooling water. An Air-Cooled Exchanger may be as small as an automobile radiator or large enough to reject the heat of turbine exhaust steam condensation from a power plant.

Details

Features:
The exchanger may be designed for two service types namely either air cooler or air condenser. The two draft types available are induced and forced draft using axial flow, propeller type fans to drive ambient air at local atmospheric pressure across a rectangular tube bundle. The hot process fluid is always considered to be inside the tubes.

Tubes may be low finned, high finned or just cylindrical. High finned tubes are usually radial, circular fins which may be tension wrapped, extruded from a bimetallic liner or embedded into the wall of the tube. Tube bundles are rectangular and may consist of one or more tube rows (or layers). A tube bundle is an assembly of tubes, headers, side frames, and tube supports. Usually the tube surface exposed to the passage of air has extended surface in the form of fins to compensate for the low heat transfer rate of air at atmospheric pressure and at a low enough velocity for reasonable fan power consumption. Tube passes may be specified as single pass or multipass. For two pass coolers, an uneven number of rows per pass may be specified. Fans may be specified as one or more per forced or induced draft type. Air-cooled exchanger determines the airflow in cubic foot per minute, outlet pressure and temperature.

Calculation Types:

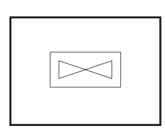
- Basic air-cooled exchanger without rating can be performed for both single and two-stream exchangers. For a single stream exchanger, air flowrate will be calculated (when not specified) and two specifications (one from the air side and one from process side) can be met. For a two-stream exchanger, one specification is met from the air and process side specifications. (please see Air-Cooled Exchanger Command Details & Air Cooled-Exchanger Optional Commands)

- Air-Cooled Exchanger Rating can be done for single-stream and two-stream exchanger types. For a single stream exchanger, one air or process stream specification can be met when geometry (physical data on fin, fan, and tubes) and inlet process conditions are fully specified. For a two-stream exchanger, the specified geometry calculates the air and process outlet conditions along with the estimated fan horsepower. Airside heat transfer and pressure drops are calculated from general correlations.

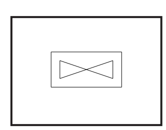
Please see the online **DESIGN II** Help, topic ***Equipment/Air Cooled Exchanger*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 3: Air Cooled Exchanger*** for more details.

Symbols

Air Cooler: Requires one inlet stream and one outlet stream.



Air Cooler (Two Stream) requires two inlet streams and two outlet streams.



Note that both symbols use the same visual image but the one stream air cooler symbol has two stream connection points. The two-stream air cooler symbol has four stream connection points. Open the **Options** menu and choose the **Show Snap Points** item.

Air Cooler Properties (Single Stream)

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

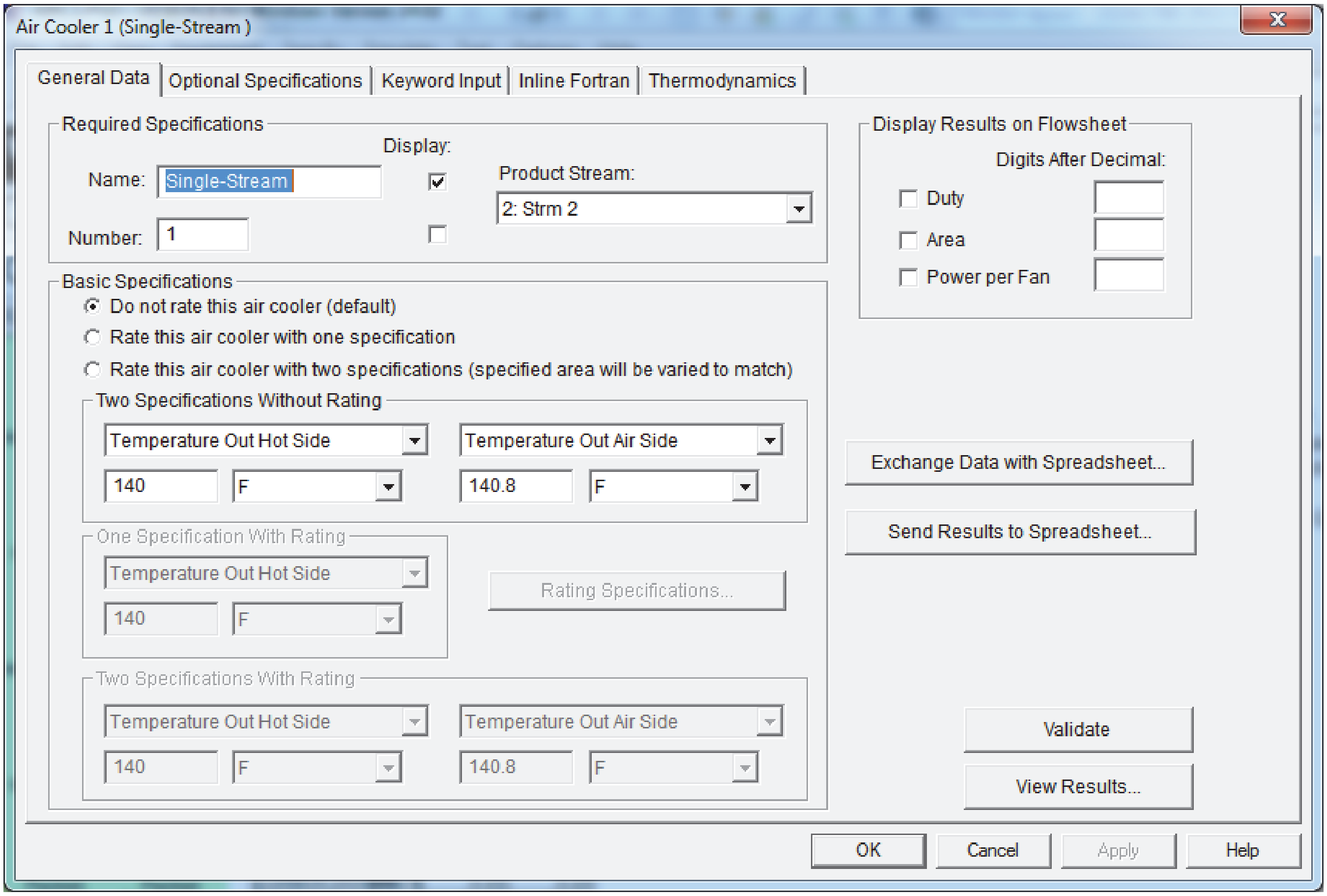


Figure 1: Air Cooler dialog (from airc1.psd)

General Data (Single Stream)

Air-Cooled Single Stream Exchanger

A single stream exchanger is defined as an exchanger with only the process stream. Air and process side stream specifications will be met, and air flowrate will be calculated when not specified.

Data Item	Description
Name/display, Number/display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Display Results on Flowsheet	You can optionally display the following results on the flowsheet: Duty Area Power per fan If you select to display one or more of these results, enter the number of digits to display to the right of the decimal, along with any text you might want to display before the results.
Basic Specifications	Select either: Do not rate this air cooler (default): If you choose this option, you can complete the Specification Without Rating section. Rate this air cooler with one specification: If you choose this option, you can complete the One Specification With Rating section ; then select the Rating Specifications... button; this displays the Rating dialog with the General Data tab selected (you can also enter data on the Tube Specifications and Fin/Fan Specifications tabs). Rate this air cooler with two specifications (specified area will be varied to match): If you choose this option, you can complete the Two Specifications with Rating section ; then

select the Rating Specifications... button; this displays the Rating dialog with the General Data tab selected (you can also enter data on the Tube Specifications and Fin/Fan Specifications tabs).

Two Specifications Without Rating

If you selected Do no rate this cooler, complete this section. Select one specification from the first list, and one from the second list.

From the first list, select either:

Temperature Out Hot Side: Temperature of the outlet hot side stream. Enter the value and select a unit.

Temperature Hot Side Change: Temperature change of the hot side stream. Enter the value and select a unit.

Temperature Approach Hot Side: Absolute temperature difference between the outlet of the hot side stream and the inlet of the air side stream. Enter the value and select a unit.

Duty: The heat subtracted from the hot side stream. Enter the value and select a unit.

From the second list, select either:

Temperature Out Air Side: Temperature of the outlet air side stream. Enter the value and select a unit.

Air Flow Rate: The amount of air flowing on the air side. Enter the value and select a unit.

One Specification with Rating

If you selected Rate this cooler with one specification, complete this section. Select one of the following:

Fan Power per Fan: The amount of power used to push air through the air side. Enter the value and select a unit.

Temperature Hot Side Change: Temperature change of the hot side stream. Enter the value and select a unit.

Temperature Out Hot Side: Temperature of the outlet hot side stream. Enter the value and select a unit.

Temperature Approach Hot Side: Absolute temperature difference between the outlet of the hot side stream and the inlet of the air side stream. Enter the value and select a unit.

Duty: The heat subtracted from the hot side stream. Enter the value and select a unit.

Temperature Out Air Side: Temperature of the outlet air side stream. Enter the value and select a unit.

Temperature Air Side Change: Temperature change of the air side stream. Enter the value and select a unit.

Air Flow Rate: The amount of air flowing on the air side. Enter the value and select a unit.

Rating Specifications...: Click this button to display the Rating dialog with the General Data tab selected (you can also enter data on the Tube Specifications and Fin/Fan Specifications tabs).

Two Specifications Without Rating

If you selected Rate this air cooler with two specifications, complete this section. Select one specification from the first list, and one from the second list.

From the first list, select either:

Temperature Out Hot Side: Temperature of the hot side stream. Enter the value and select a unit.

Temperature Hot Side Change: Temperature change of the hot side stream. Enter the value and select a unit.

Temperature Approach Hot Side: Absolute temperature difference between the outlet of the hot side stream and the inlet of the air side stream. Enter the value and select a unit.

Duty: The heat subtracted from the hot side stream. he output for the exchanger will report a positive duty for heat added to the first inlet (shell side) stream and a negative duty for heat removed from the first inlet (shell side) stream. Enter the value and select a unit.

From the second list, select either:

Temperature Out Air Side: Temperature of the outlet air side stream. Enter the value and select a unit.

Temperature Air Side Change: Temperature change of the airside stream. Enter the value and select a unit.

Air Flow Rate: The amount of air flowing on the air side. Enter the value and select a unit.

Send Results to Spreadsheet, Exchange Data with Spreadsheet, Validate, View Results	Refer to the Chapter 3: Basics for details.

General Data (Two Stream)

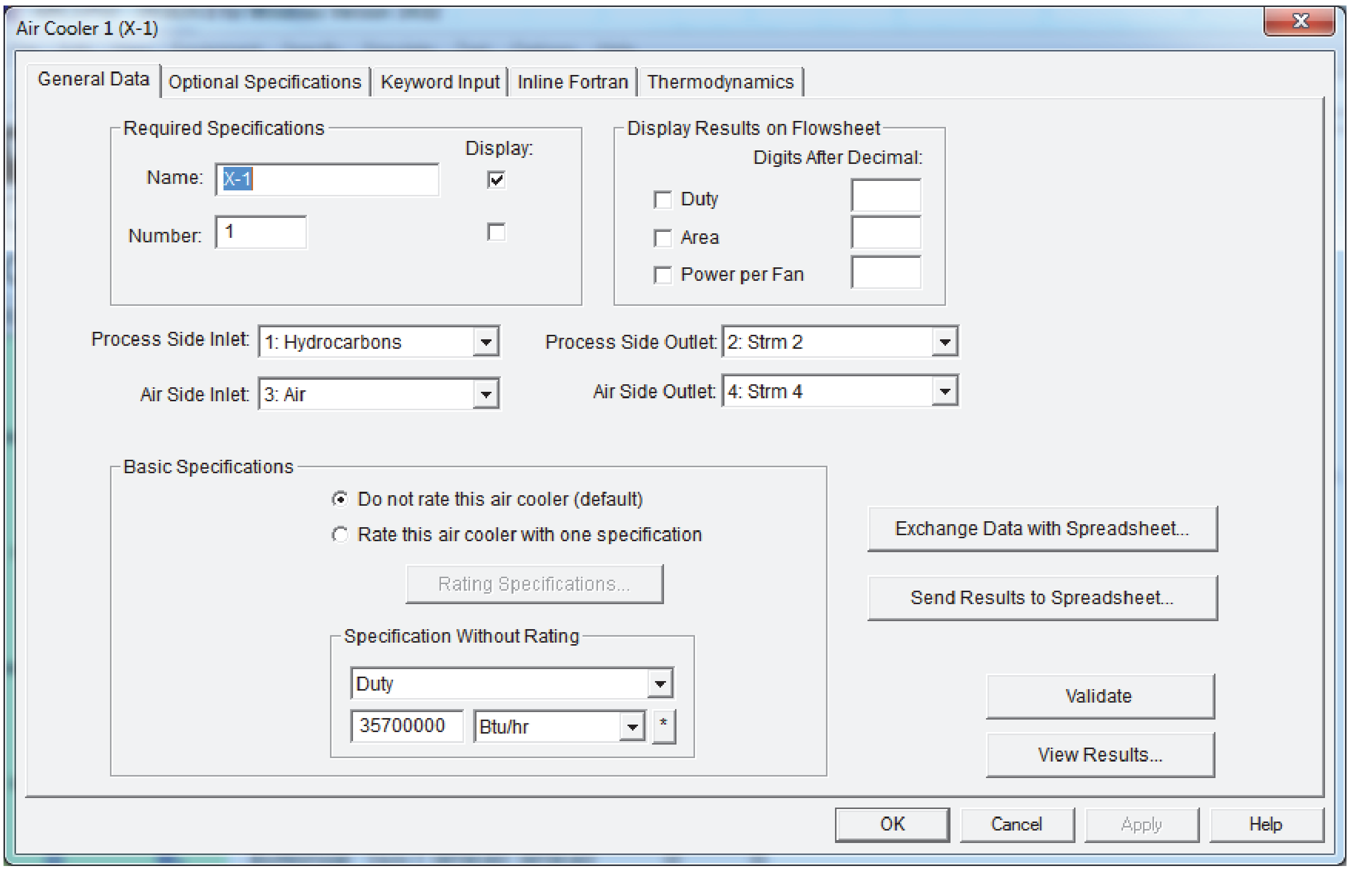


Figure 2: Air Cooler Dialog (Two-Stream) (from airc3.psd)

A two stream exchanger is defined as an exchanger with both the process and the air streams specified with temperature, pressure, and flow. One specification from the list below for the air (or) process side stream will be met. The specifications are:

Data Item	Description
Name/display, Number/display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Display Results on Flowsheet	You can optionally display the following results on the flowsheet: Duty Area Power per fan If you select to display one or more of these results, enter the number of digits to display to the right of the decimal, along with any text you might want to display before the results.

Process Side Inlet/Air Side Inlet	Select the streams to use for the process side and air side inlets.
Process Side Outlet/Air Side Outlet	Select the streams to use for the process side and air side outlets.
Basic Specifications	Select either: Do not rate this air cooler (default): If you choose this option, you can complete the Specification Without Rating section. Rate this air cooler with one specification: If you choose this option, you can select the Rating Specifications... button to display the Rating dialog with the General Data tab selected (you can also enter data on the Tube Specifications and Fin/Fan Specifications tabs).
Specification Without Rating	If you selected Do no rate this cooler, complete this section. Select one specification from the first list, and one from the second list. From the first list, select either: Temperature Out Hot Side: Temperature of the outlet hot side stream. Enter the value and select a unit. Temperature Hot Side Change: Temperature change of the hot side stream. Enter the value and select a unit. Temperature Approach Hot Side: Absolute temperature difference between the outlet of the hot side stream and the inlet of the air side stream. Enter the value and select a unit. Duty: The heat subtracted from the hot side stream. Enter the value and select a unit. Temperature Out Air Side: Temperature of the outlet air side stream. Enter the value and select a unit. Air Flow Rate: The amount of air flowing on the air side. Enter the value and select a unit.
Send Results to Spreadsheet, Exchange Data with Spreadsheet, Validate, View Results	Refer to Chapter 3: Basics for details.

Rating Data

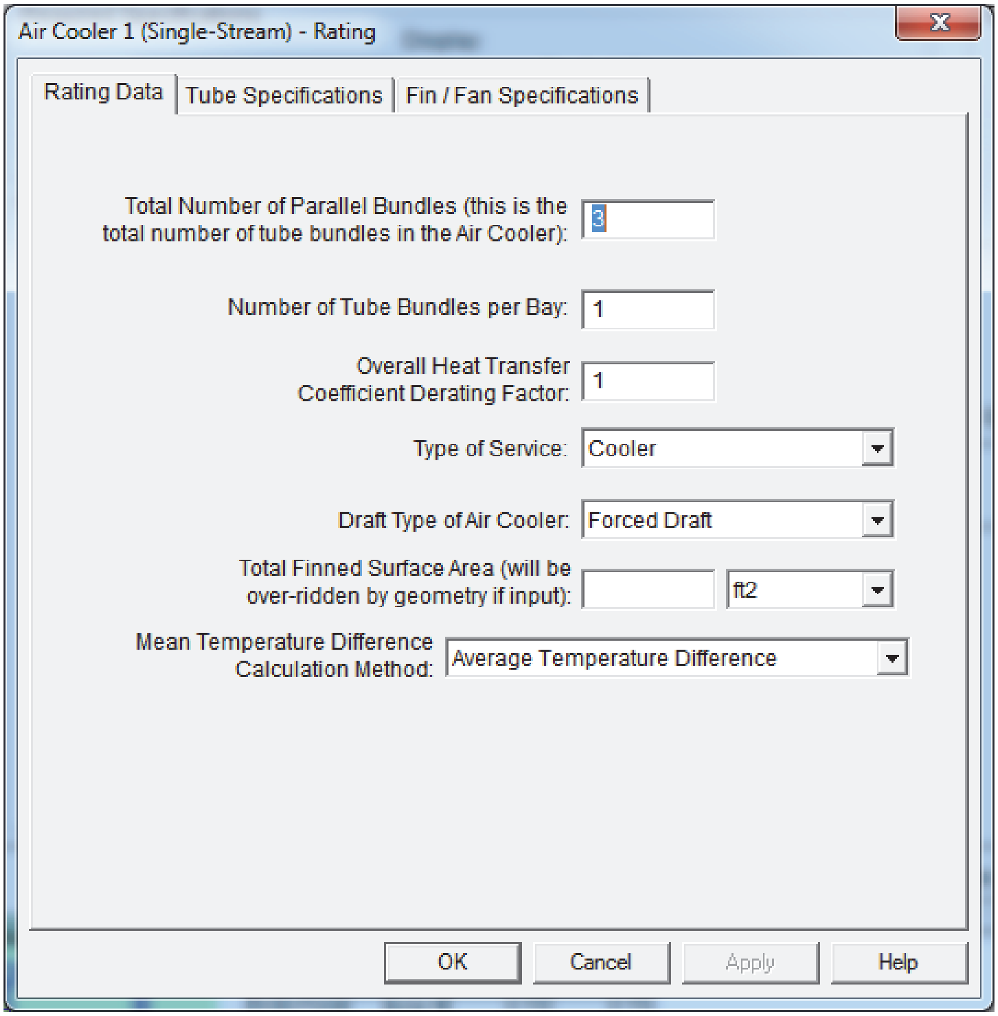


Figure 3: Rating Data (from airc2.psd)

Data Item	Description
Total Number of Parallel Bundles	Enter a value. The default is 1.
Number of Tubes Bundles Per Bay	Enter a value. The number of bays is calculated as the ratio of number of parallel bundles to number of bundles per bay. The default is 1.
Overall Heat Transfer Coefficient Derating Factor	Enter a value to specify "de-rating" factor for each stream that will be multiplied with the calculated heat transfer coefficient. The default is 1.
Type of service	This is the type of service performed by the air-cooled exchanger. If the choice is NOT entered flash of phase condition in and out determine which type of air-cooler is chosen. TWO choices are available: 1 (default) Cooler, single phase, process side 2 Condenser, two phase, process side
Draft type of air cooler	This is the draft type of air-cooled exchanger. TWO choices are available: 1 (default) forced draft coolers 2 for induced draft coolers
Total Finned Surface Area	Enter the value and select a unit. If a number of tubes is specified, the area can be calculated from tube geometry (given or defaulted), which then overrides this value.

Mean Temperature Difference Calculation Method	Select one of the following: Log Mean Temperature Difference (default) : the log mean temperature difference between wall and fluid. Average Temperature Difference: the difference between the wall and the fluid temperature (this is normally the most stable method). Average Cube Root Mean Difference: the average cube root mean temperature difference between wall and fluid.

Tube Specifications

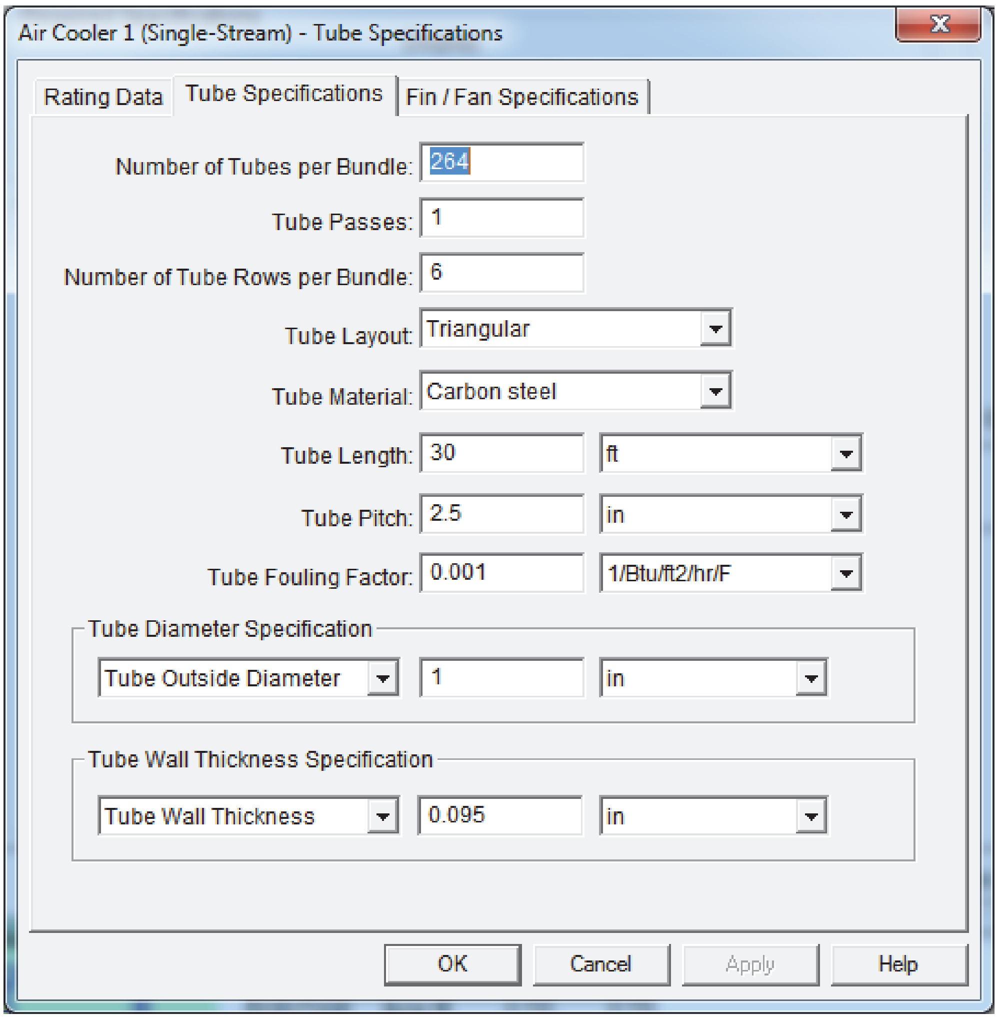


Figure 4: Tube Specifications (from airc2.psd)

Data Item	Description
Number of Tubles Per Bundle	Enter the number of tubes per bundle of the air-cooled exchanger. The default is 228.
Tube Passes	Enter the number of tube passes per shell. The default is 1.
Number of Tube Rows Per Bundle	Enter the number of tube rows per bundle of the air-cooled exchanger. The default is 1.
Tube Layout	Enter your choice for tube layout. Options are: **SQU**are (default), **ROT**ated **TRI**angular, **ROT**ated **SQU**are or **TRI**angular.
Tube Material	**ADM**iralty brass **ALU**minum **CAR**bon steel **COP**per **COP**per nickel **70**/30 **COP**per nickel **90**/10

MONel
NICkel 200
RED brass
STAinless steel 304
ALLoy 825
CARbon **MOL**y Steel
STAinless steel **214**
TITanium
Glass
PVC

Tube Length	Enter the length of the tubes. Default is 16 feet.
Tube Pitch	Enter the tube pitch. Default is 0.9375 inches. TUBe PASses is also used by the rating calculation. The default value for the number of TUBe PASses is one. Entry of a value for this command affects both calculations.
Tube fouling factor	Enter the fouling factor for the tubes. Default is 1E-03 1/BTU/FT2/HR/ F. Other Units allowed are 1/KCAL/M2/HR/C, 1/KJ/M2/SEC/K, 1/CAL/M2/HR/C.
Tube Diameter specification	
Tube Inside Diameter	Enter the inside diameter of the tubes. If no value is entered, it will be calculated, when TUBe BWG is specified.
Tube Outside Diameter	Enter the outside diameter of the tubes. Default is 0.75 inches.
Tube Wall Thickness specification	
Tube BWG	Enter the Birmingham Wire Gauge for tube wall thickness. Default is 12.
Tube wall thickness	Enter the wall thickness of the tubes. If no value is entered, the program will calculate it.

Fin/Fan Specifications

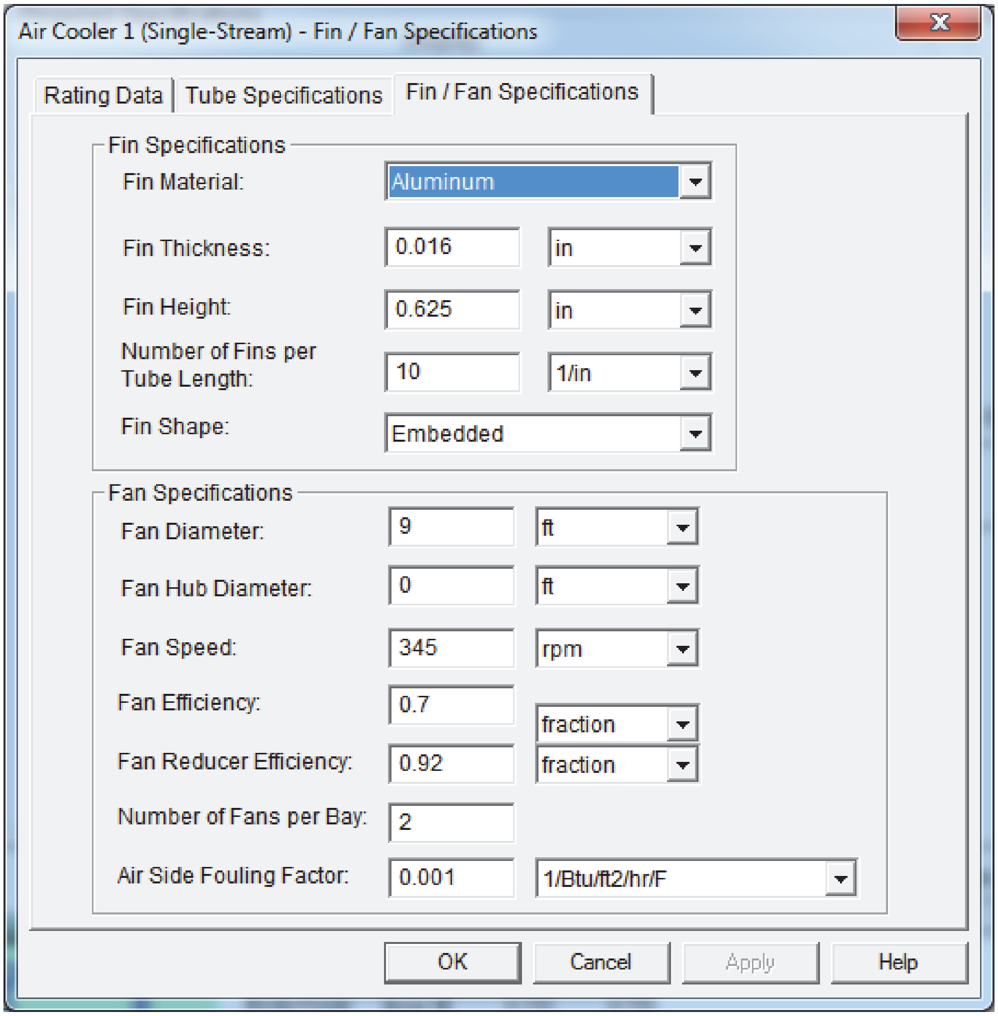


Figure 5: Fin/Fan Specifications (from airc2.psd)

Data Item	Description
Fin material	Enter the choice of fin material. (default is **ALU**minum). Choices are listed below. If you cannot find the specific tube material in the Heat Exchanger: Tube Metallurgy Table, select the material which has an average thermal conductivity closest to the desired material. **ADM**iralty brass **ALU**minum **CAR**bon steel **COP**per **COP**per nickel **70**/30 **COP**per nickel **90**/10 **MON**el **NIC**kel 200 **RED** brass **STA**inless steel 304 **ALL**oy 825 **CAR**bon **MOL**y Steel **STA**inless steel **214** **TIT**anium Glass PVC
Fin Thickness	Enter this command to specify the thickness of air cooler tube fins. The default is 0.016 inches.

Fin Height	Enter this command to specify the height of the air cooler fins. Calculated from the outside diameter of the pipe and fin (Fin height = 1/2 fin outside diameter - 1/2 pipe outside diameter).
Number of fins per tube length	Enter the number of fins per inch of air-cooled exchanger tube length. The default for fins/length is 132 fins per foot (11 fins per inch)
Fin Shape	Enter the choice of fin shape. Choices are listed below: **STR**aight (default), **EXT**ruded, **TRI**angular or **EMB**edded
Fan diameter	Enter the diameter (from blade tip to blade tip) of the fan. The default is 7 ft.
Fan hub diameter	Enter the diameter of the fan hub (i.e. the central cylinder to which the fan blades are attached and through which no flow occurs). The default value for diameter is zero feet, implying that the crosssectional area taken up by the hub is negligible.
Fan Speed	Enter this command to specify the fan speed in rpm.
Fan Efficiency	Enter the fan efficiency as a fraction. Default is 0.7.
Fan reducer efficiency	Enter the fan reducer efficiency as a fraction. Default is 0.92.
Number of fans per bay	Enter the number of separate fans in the air- cooled exchanger.
Airside Fouling Factor	Enter the fouling factor for the airside. Default is 1E-03 1/BTU/FT2/HR/ F. Other Units allowed are 1/KCAL/M2/HR/C, 1/KJ/M2/SEC/K, 1/CAL/M2/HR/C.

Air Cooler (Single Stream and Two Stream) Optional Specifications

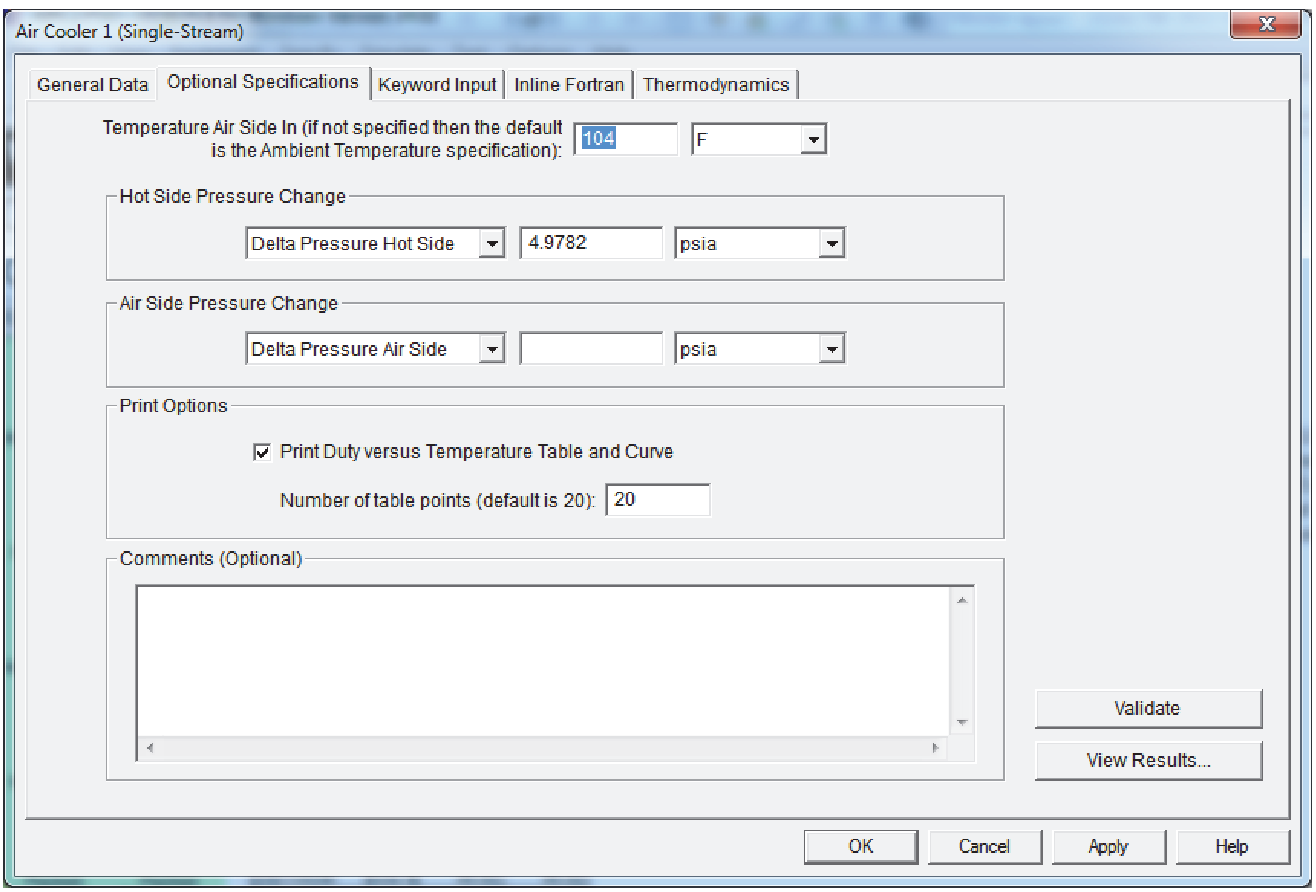


Figure 6: Optional Specifications tab from the Air Cooler dialog (from airc2.psd)

Data Item	Description
Temperature Air Side In	Enter the inlet temperature for the air stream. If not entered default ambient temperature will be used. Default of Temperature of ambient is 77F.
Hot Side Pressure Change	Select one of the following; then enter a value and select a unit: Delta Pressure Hot Side Pressure Out Hot Side
Air Side Pressure Change	Select one of the following; then enter a value and select a unit:

	Delta Pressure Air Side Pressure Out Air
Print Duty versus Temperature Curve	Select this choice to plot duty versus temperature curves for the air cooled exchanger. The curves are a useful tool for evaluating exchanger design. These curves are based on end-point temperature results and counter current flow. Since air coolers have cross flow, the minimum temperature approach will not be accurate and may actually show cross-overs which DO NOT actually exist. In addition, bubble points and dew points are marked when vaporization or condensation occurs for hot streams. Number of table points: Enter the number of table points to use for creating the curve; the default is 20.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide. Use **the Inline Fortran** tab to enter this information.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II Reference Guide.

Air-Cooled Exchanger Examples

There are several sample flowsheets in “Chapter 9: Air Cooled Exchanger Samples - c:\designii\samples\equipmnt\aircoolr” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.3: Amine Absorber/Regenerator

General

The Amine Column (Absorber or Regenerator) makes use of a rate model for the rigorous simulation of absorbers, flash-gas reabsorbers and regenerators used in mixed Amine-based (MEA, DEA, MDEA, DGA, and DIPA) gas treating processes. This truly rigorous model treats the separation as a mass transfer and/or chemical reaction rate controlled process and calculates the actual separation being achieved on each tray, or group of trays without recourse to stage efficiencies or HETP data. Detailed tray construction is used in the model. This module can be used for design, modification, or optimization of rigorous amine distillation columns. With the use of the proper option, the operation of existing units can be duplicated within several percent of actual plant data.

Details

You can use the amine absorber column or regenerator column for designing, modifying, or optimizing rigorous amine distillation columns. The amine column makes use of a rate model for the rigorous simulation of absorbers, flash-gas reabsorbers and regenerators used in Amine-based (MEA, DEA, MDEA, DGA, and DIPA) gas treating processes.

This truly rigorous model treats the separation as a mass transfer and/or chemical reaction rate controlled process and calculates the actual separation being achieved on each tray, or group of trays without recourse to stage efficiencies or HETP data. Detailed tray construction is used in the model. The thermodynamic models used are available in the Thermodynamics: Mixed Amine section of the DESIGN II help. With the use of the proper option, the operation of existing units can be duplicated within several percent of actual plant data.Stream data are available from the simulation for all streams within columns including all gas and liquid streams flowing between trays. Because Amine Column deals with real physical trays, the compositions, flow rates and physical properties reported are those that should actually exist at various points within the columns.

Amine Column accesses thermodynamic and physical properties data bases that allow it to rigorously model carbon dioxide, hydrogen sulfide removal and selective hydrogen sulfide removal from a wide variety of gas streams using MEA, DEA, DGA, DIPA and MDEA. There are no inherent limitations on solvent strengths or acid gas loadings.

Flowsheet

Inlet: Two inlet feed streams are mandatory for ABSorber modules and one additional side feed is allowed for the second absorber module. For regenerator module only one feed is allowed.

Outlet: Two material streams should leave for both the absorber and regenerator modules. One sidedraw is allowed for the regenerator module only.

Tray Ordering Convention and Input Specifications

The Amine Column ordering convention is that trays are numbered from the top of the column down (not including reboiler or condenser) and that locations within packed sections are also measured by distance from the top.

There are two types of input specifications, namely, stream specifications and equipment specifications. Amine Column input is restricted to equipment specifications only. Some data are pressure of column, number of trays, pressure drop, reboiler duty, bottoms flow from reboiler, and reboiler duty are equipment specifications. Solution circulation rates are given as part of the flowsheet specifications.

Amine Column begins with the basic column and determines the existence of a reboiler and / or a condenser. If a condenser is present, its temperature is a necessary specification.

Column Types
If the column is TRAYED, the following specifications must be made:

- total number of actual trays.
- either pressure on one tray and pressure drop per tray (specified or calculated) or pressure on two trays. If pressure on two trays is specified, a linear pressure profile is assumed regardless of changes in tower diameter or tray geometry.
- tray type (VALve (or) BUBble (or) SIEve), spacing and weir height in each section of the column where these differ.
- number of tray passes in each section of the column where this differs
- active tray area (as actual total area, actual area per pass, or percent of tower area) for each section of the column where these differ.
- tower diameter (actual or as percent of flood) for each section of the column where these specifications differ.

If the column is PACKED, the following specifications are necessary:

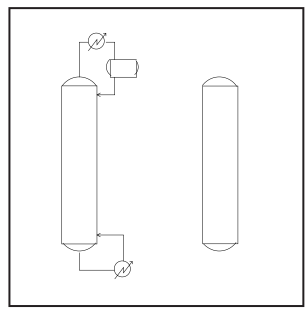
- total packed depth - either pressure at one depth and pressure drop per unit depth (specified or calculated) or pressure at two depths. If pressure at two depths is specified, a linear pressure profile is assumed throughout the column regardless of changes in tower diameter and packing parameters.
- for dumped packing - the nominal size, packing factor and dry area per unit volume for each section of the column where these are different.
- tower diameter (actual or as percent of flood) for each section of the column where this specification differs.

For all types of columns it is necessary to specify exact column entry and exit points for all streams.

Please see the online **DESIGN II** Help, topic ***Equipment/Amine Column*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 4: Amine Column*** for more details.

Symbols

The Amine Column unit module has two symbols: Amine Regenerator and Amine Absorber.



The Absorber requires a liquid feed stream to the top left or top right of columns. A liquid product stream must be connected to the top of the vessel. A liquid product stream must be connected to the bottom of the vessel. A vapor feed stream must be connected to the bottom left or right of the column. A side feed stream can be connected to the column.

The Regenerator requires a vapor product stream be connected to the top of the condenser. A bottom liquid product stream must be connected to the bottom of the column. A side draw stream may be connected on either side of the column.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

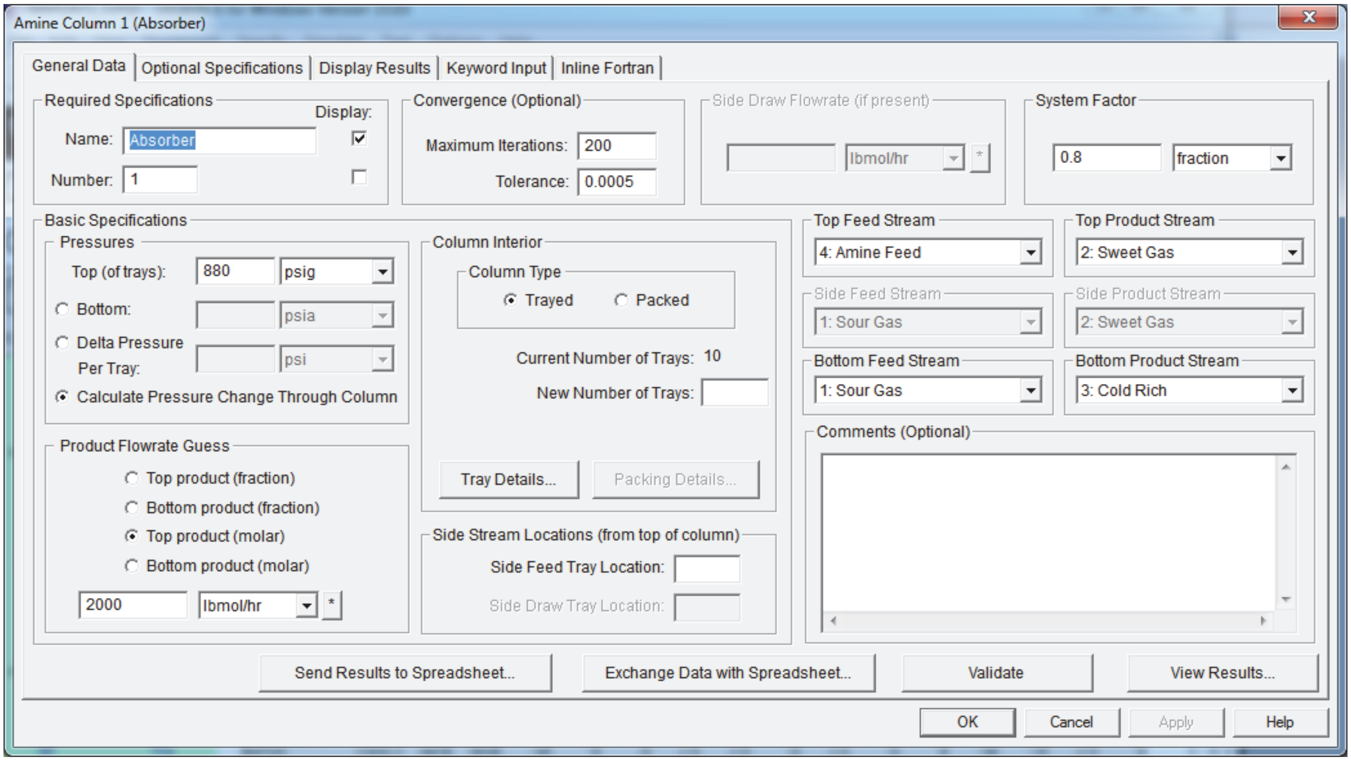


Figure 1: Amine Absorber Dialog (from mxplnt1.psd)

General Data

Data Item	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Maximum Iterations	The default value is normally adequate but can be overridden. Enter the maximum number of iterations to use.
Tolerance	The default value is normally adequate but can be overridden. Enter the maximum acceptable NORM for the column. The NORM is the error in the tray heat and material balance on a fractional basis squared.
Pressures	
TOP of tray (P units)	The top tray/packed section pressure will be used as the pressure of the top product. Enter a value and select a unit. Then, choose either Bottom, Delta Pressure Per Tray, or Calculate Pressure Change Through Column.
Bottom	If you choose this option, enter the bottom tray pressure. The bottom tray/packed section pressure will be used as the pressure of the bottom product. Enter a value and select a unit.
DELta Pressure per TRAy (P units) =	If you choose this option, enter the pressure drop from the bottom product pressure to the top tray or packed section pressure. Enter a value and select a unit.
Calculate Pressure Change Through Column	If you choose this option, the pressure drop across the trays or packed section depth will automatically be calculated.
Column Type	Select the Column type, either: Trayed Packed If you select Trayed: Current Number of Trays: This item reports to you the number of trays you have specified for the column. If you have not entered the number of trays, the space will be blank.

	New Number of Trays: Enter the number of theoretical trays in the column. When you change the number of trays a third field appears. Click the Tray Details button to enter Tray Specifications. If you select Packed: Current Number of Packing Sections: This item reports to you the number of packing section you have specified for the column. If you have not entered the number of sections, the space will be blank. New Number of Sections: Enter the number of sections in the column. When you change the number of sections a third field appears. Click the Packing Details button to enter Packing Specifications.
Product Flowrate Guess	Select one of the following flow rate guesses: Top product (fraction) Bottom product (fraction) Top product (molar) Bottom product (molar) Then, enter a corresponding value and select a unit.
Side Stream Locations (from top of column)	If you selected Trayed: Side Feed Tray Location: Enter a value. Side Draw Tray Location: Enter a value. If you selected Packed: Side Feed Depth: Enter a value and select a unit. Side Draw Depth: Enter a value and select a unit. To change the base quantity and time units, click here *.
Side Draw Flowrate (if present)	If the column has a side draw, enter a product flowrate value and select a unit. To change the base quantity and time units, click here *.
System Factor	Specify the foaming characteristics in the column. Enter a value and select a unit.
Top Feed Stream/Top Product Stream/Side Feed Stream/Side Product Stream/ Bottom Feed Stream/ Bottom Product Stream	Select the desired stream to use for each type of stream.
Send Results to Spreadsheet/ Exchange Data with Spreadsheet/Validate/View Results	See Chapter 3: Basic for details.

Main Specifications

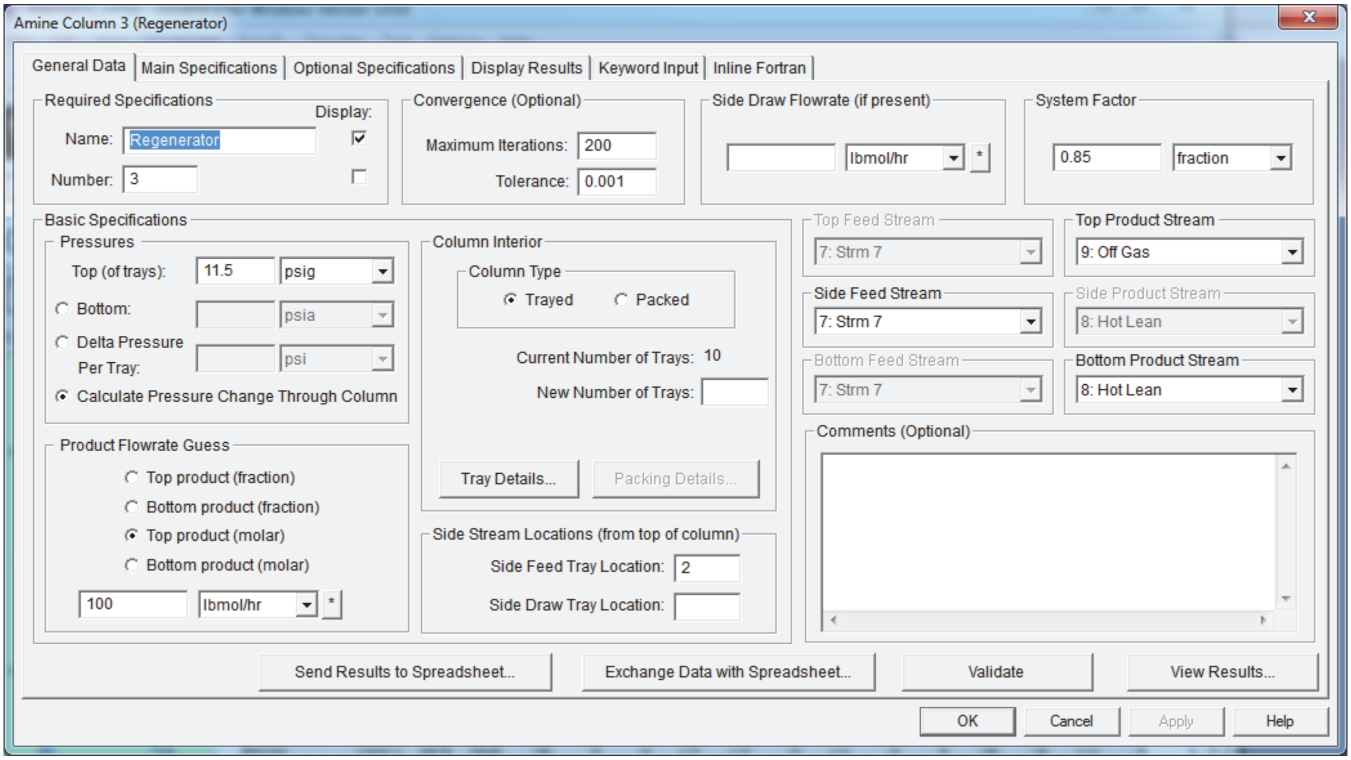


Figure 2: Main Specifications tab (from mxplnt1.psd)

Data Item	Description
Column Specification Choice	Select one of the following: Reboiler Duty and Condenser Temperature Reflux Ratio and Product FlowRate Guess (See General Data tab) Reflux Ratio and Condenser Temperature
Reflux Ratio	If you selected either Reflux Ratio and Product FlowRate Guess or Reflux Ratio and Condenser Temperature, enter a reflux ratio. Enter the reflux to total overhead product molar ratio. The total overhead product is vapor distillate plus liquid distillate.
Condenser	
Pressure	For any column selection choice, enter the condenser outlet pressure value and select a unit.
Temperature	If you selected Reboiler Duty and Condenser Temperature or Reflux Ratio and Condenser Temperature, enter the condenser temperature value and select a unit.
Reboiler	
Pressure	For any column selection choice, enter the reboiler outlet pressure value and select a unit.
Duty	If you selected Reboiler Duty and Condenser Temperature, enter the reboiler duty value and select a unit.

Tray Details

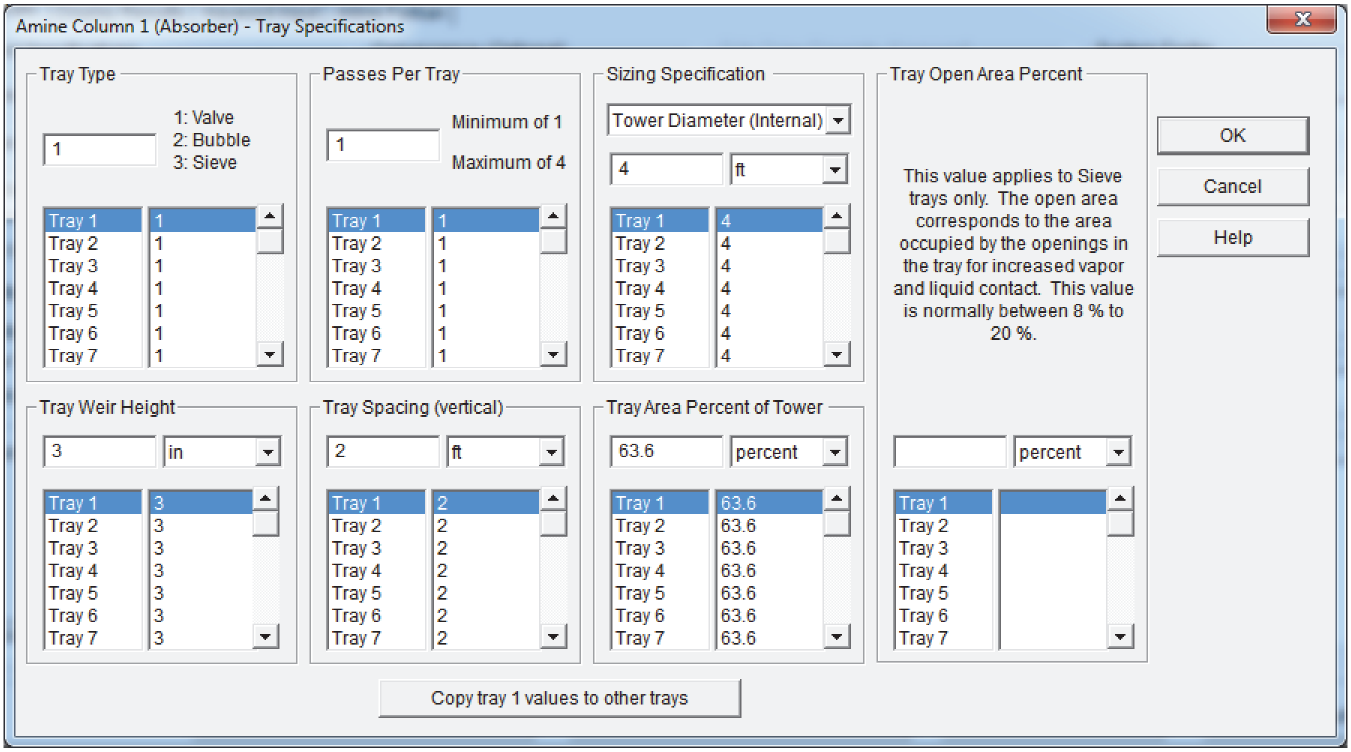


Figure 3: Tray Specifications Dialog (from mxplnt1.psd)

Data Item	Description
Tray Type	Three choices are available in tray types. They are VALve, BUBble, and SIEve.
Passes PerTray	Enter the number of passes usually needed to control excessive liquid gradient. Available choices are 1, 2, 3, and 4
Sizing Specifications	Enter the tower or column actual internal diameter in feet. Multiple diameters can be specified for each tray.
Tray Weir height	Enter the WEIr HEIght to be used along with TRAy SPAcing.
Tray Spacing	Enter the tray spacing to be used along with TRAy WEIr height.
Tray Area percent of Tower	Enter the tray area specified as percent of the tower or column area.
Tray Open Area Percent	Enter the tray open area percent. This applies for SIEve trays only, where the open area corresponds to number and area occupied by the openings in the tray for increased vapor and liquid contact. Normally the tray open area percent is anywhere between 8 to 20 %.

Packing Details

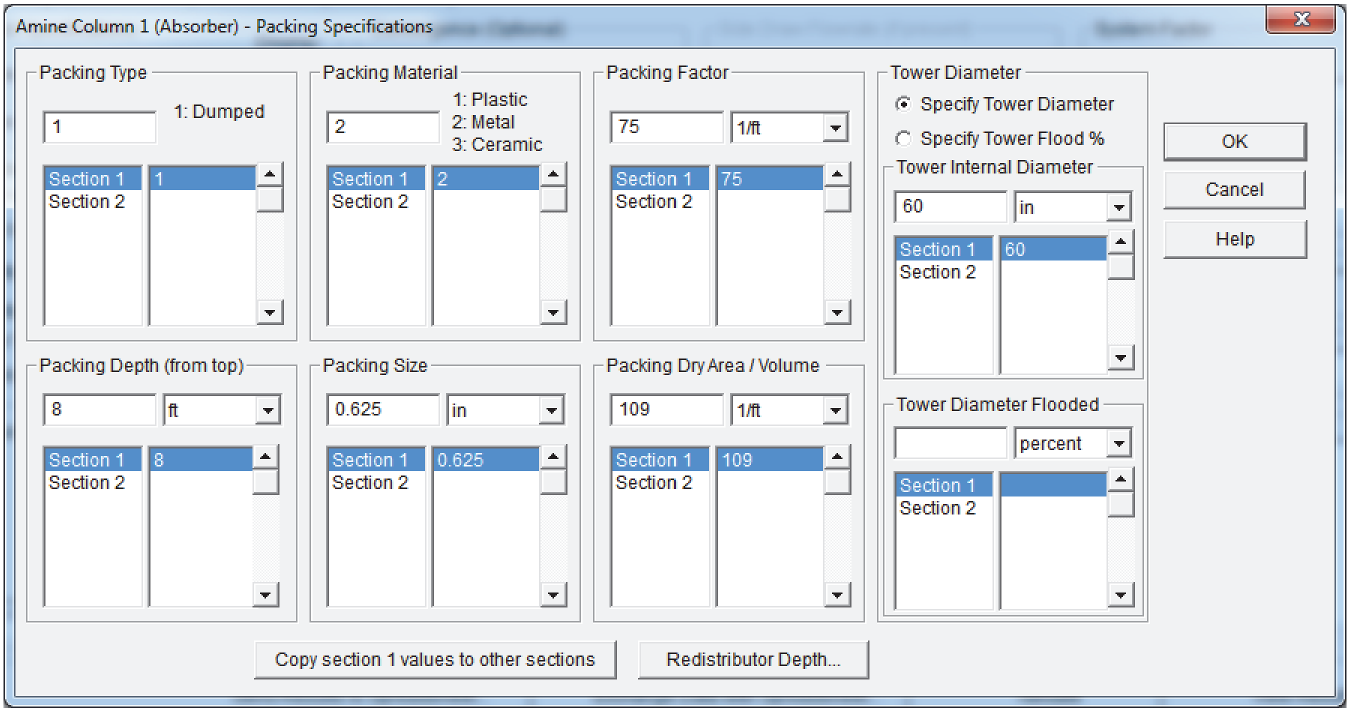


Figure 4: Packed Details (from mxpkabsf.psd)

The method is general for dumped packing only. All commands are needed to design the packed sections within the column except otherwise noted in specific commands for its alternative use.

Data Item	Description
Packing Type	Enter the type of packed material as DUMPed per section.
Packing Material	Three choices are available in packing material. They are PLASTIC (1), METAL (2) and CERAMIC (3).
Packing Factor	Enter the packing factor per section to specify the degree of packing arrangement (for dumped packing).
Pack Depth from Top	Enter the packing depth per section from the top of the column. Maximum of 10 packing sections are allowed.
Packing Size	Enter the packing material's size per packing section.
Packing dry area	Enter the packing dry surface area per unit volume for each packing section. This accounts for the porous nature of the packing material and defines the amount of available active surface for adequate liquid/ vapor contact. The dimensional units are area/ volume (or) 1/ length units.
Tower Diameter	Enter the TOWer actual internal DIAmeter per section. Should not be specified if TOWer DIAmeter FLOOD is used.
Tower Diameter Flood	Enter either the TOWer DIAmeter FLOODing percent per section. Should not be specified if TOWer actual internal DIAmeter is used.

Redistributors

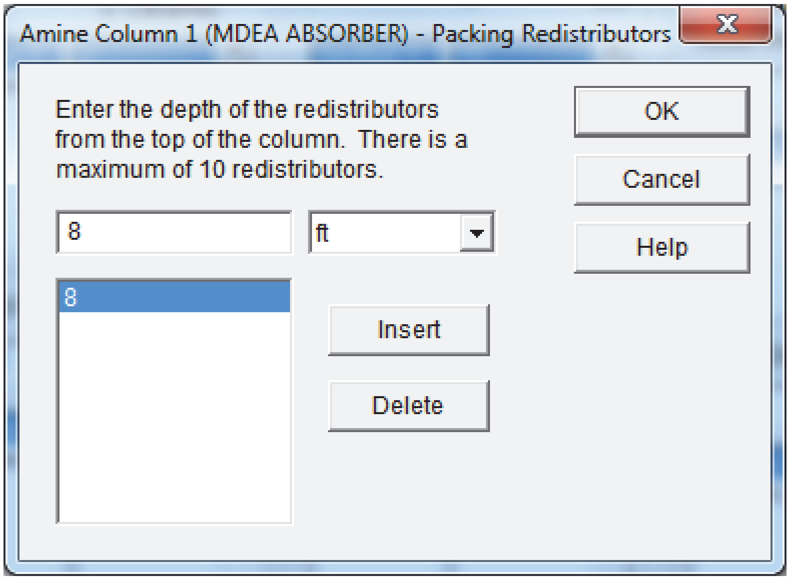


Figure 5: Redistributors Dialog (from mxpkabsf.psd).

Data Item	Description
Redistributors Depth	Enter the depth of the redistributors from the top of the column in length units.A maximum of 10 distributors are allowed. Redistributors are not connected to a section. It serves to collect the liquid running down from the wall that has coalesced at certain sections of the column ; then re-distributes it to establish a uniform pattern of liquid/gas contact. Enter a value and select a unit; then click the Insert button. To remove a depth, click on it in the list then click the Delete button.

Display Results

You can choose to display a variety of results on the spreadsheet (e.g. trays, reflux ratio, etc.), and specify the digits after the decimal and any leading text label.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Amine Column Examples

There are several sample flowsheets in “Chapter 10: Amine Column Samples - c:\designii\samples\equipmnt\aminecol” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.4: Batch Distillation

General

The Batch Distillation module provides unequaled flexibility and power with the following features:

- Batch and Continuous Feeds
- Total or Partial Condenser
- Single or Multiple Product Collection Tanks for Each Product Draw
- Rigorous Kinetic Modeling of Reactions
- User Specified Events or Change Column Operations
- Variable Feed and Product Draw Rates
- Total Reflux Conditions Specification
- Cumulative or Instantaneous Product Composition Specifications
- Liquid Hold Up Profile Specification
- Pressure Profile Specification
- Conditional Logic Available to Control Feeds, Products and Events
- Converges Column to a large Variety of Heat and Mass Balance Specifications
- Step Functions
- Ramp Functions
- Several Integration Methods Available
- User Specified Integration Step Size
- Flexible Reporting Features
- Flexible Plotting Features

Details

The terms and definitions for Batch Distillation are for the most part consistent with DESIGN II; however, two exceptions bear mentioning. Because of the unsteady state nature of batch distillation, streams should be considered to be tanks which serve to hold feeds or collect products. Also, keywords are grouped in specific sections within the Batch unit operation unlike other DESIGN II unit operations which have no order restriction.

Please note the following limitations and assumptions regarding Batch calculations and output.

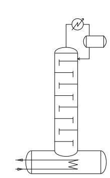
1. The continuous and batch feeds are always at the bubble point before integration begins. The heat duties printed in the output do not reflect the duty required to bring the material to the bubble point.
2. The component flow rates in the stream summary and detailed stream printout represent the cumulative flows into the product tanks. The temperature reported in the stream summary and detailed stream print consistent with the cumulative heat balance of the product tank.
3. The product stream flow rates for DESIGN II batch product streams are set to the entire contents of product tank flowing in a one hour time period. If a product stream is to be used to feed another down stream unit operation, the STReam MANipulator can be used to set the flow at the desired rate and a HEAt EXChanger should be used to account for any heat losses is located along with the column configuration commands.

Please see the online **DESIGN II** Help, topic ***Equipment/Batch Distillation*** or the **DESIGN II Unit Module** Reference Guide ***Chapter 5: Batch Distillation*** for more details.

Symbols

The Batch Distillation unit module has one symbol:

A batch feed stream must be connected to the bottom product tank. A vapor product stream must be connected to the top of the condenser. If partial condenser, a liquid product stream must be connected to the bottom of the condenser.



Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

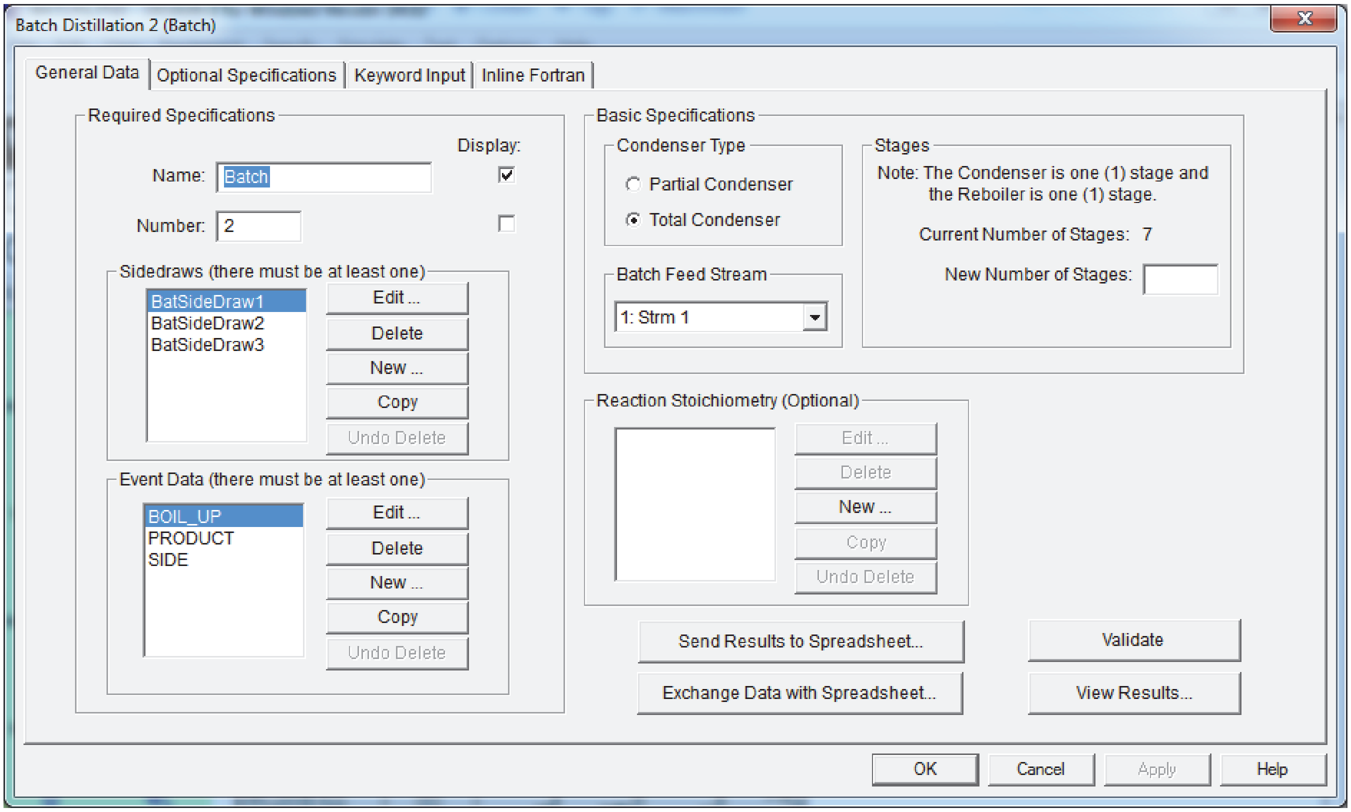


Figure 1: Batch Distillation Dialog (from batch1.psd)

General Data

Data Item	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Sidedraws	Enter sidedraws to use. Click the New button to display the Batch Side Draw dialog that you can use to create new sidedraw specifications. Select an existing sidedraw and click the Edit button to modify the sidedraw specifications. Select an existing sidedraw and click the Delete button to remove that sidedraw. Select an existing sidedraw and click the Copy button to replicate the sidedraw. You can then use the Edit function to make any minor changes to the copied sidedraw.
Condenser Type	Select either partial condenser or total condenser If you select Partial Condenser, overhead product is a vapor stream. Liquid distillate is defined as side draw. Liquid draw cannot be located on stage number 1. Therefore a liquid distillate for a partial condenser is defined as a liquid side draw located on stage 2.
Batch Feed Stream	Select a stream from the list to use as the feed.
Stages	View the current number of stages and enter the total number of theoretical stages including any condensers and reboilers. **NOTE:** A condenser is one stage and a reboiler is one stage.
Event Data	At least one event is mandatory and represents the start of the distillation. Any subsequent events begin immediately after the previous event terminates. Enter event data to use. Click the New button to display the Batch Distillation Event- Specifications dialog that you can use to create new event data specifications. Select an existing event data and click the Edit button to modify the event data specifications. Select an existing event data and click the Delete button to remove that event data. Select an existing event data and click the Copy button to replicate the event data. You can then use the Edit function to make any minor changes to the copied event data.

Reaction Stoichiometry	Enter reaction stoichiometry to use. Click the New button to display the Batch Side Draw dialog that you can use to create new reaction specifications. Select an existing reaction and click the Edit button to modify the reaction specifications. Select an existing reaction and click the Delete button to remove that reaction. Select an existing reaction and click the Copy button to replicate the reaction. You can then use the Edit function to make any minor changes to the copied reaction.
Send Results to Spreadsheet/ Exchange Data with Spreadsheet/Validate/View Results	See Chapter 3: Basic for details.

In the current release a Liquid draw cannot be located on stage number 1. Therefore a Liquid distillate for a partial condenser is defined as a liquid side draw located on stage 2. The Murphree tray efficiency for stage 2 may be set to 0.0 so that the liquid distillate is thermodynamically consistent with the liquid from stage 1. You should therefore include an extra stage when using this option.

Sidedraws

When you select the New button under Sidedraws on the General Data tab, the Batch Side Draw dialog appears.

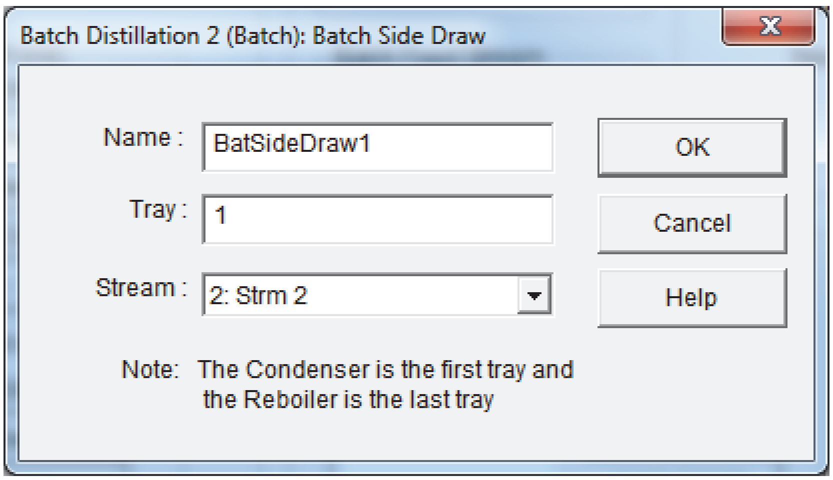


Figure 2: Batch Side Draw Dialog (from batch1.psd)

Data Item	Description
Name	Enter a name for the sidedraw.
Tray	Enter the tray number for the sidedraw.
Stream	Open the drop down list and select the stream to use as the sidedraw.

Event Data

When you select the New button under Event Data on the General Data tab, the Event - Specifications dialog appears.

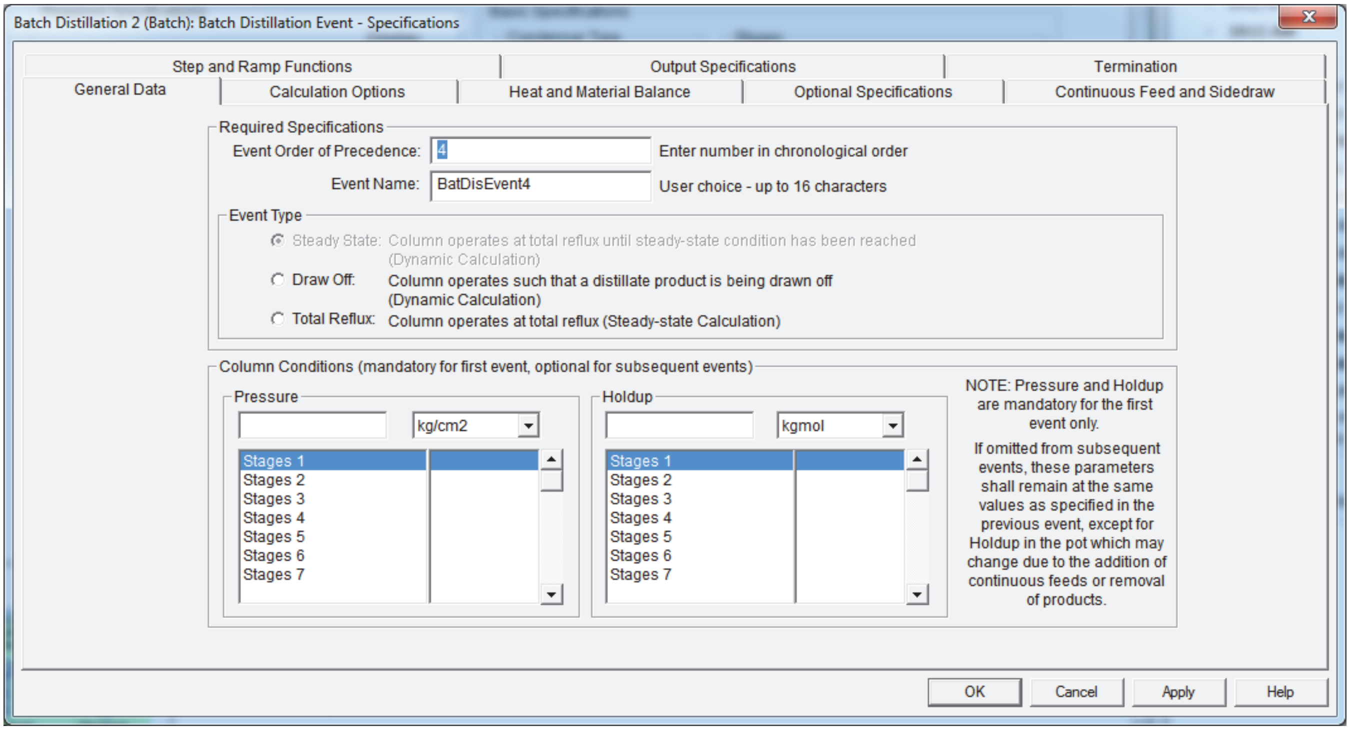


Figure 3: Batch Distillation Event - Specifications Dialog (from batch1.psd)

At least one Event is mandatory and represents the start of the distillation. Any subsequent event begins immediately after the previous event terminates.

Reaction

When you select the New button under Reaction Stoichiometry on the General Data tab, the Reaction dialog appears.

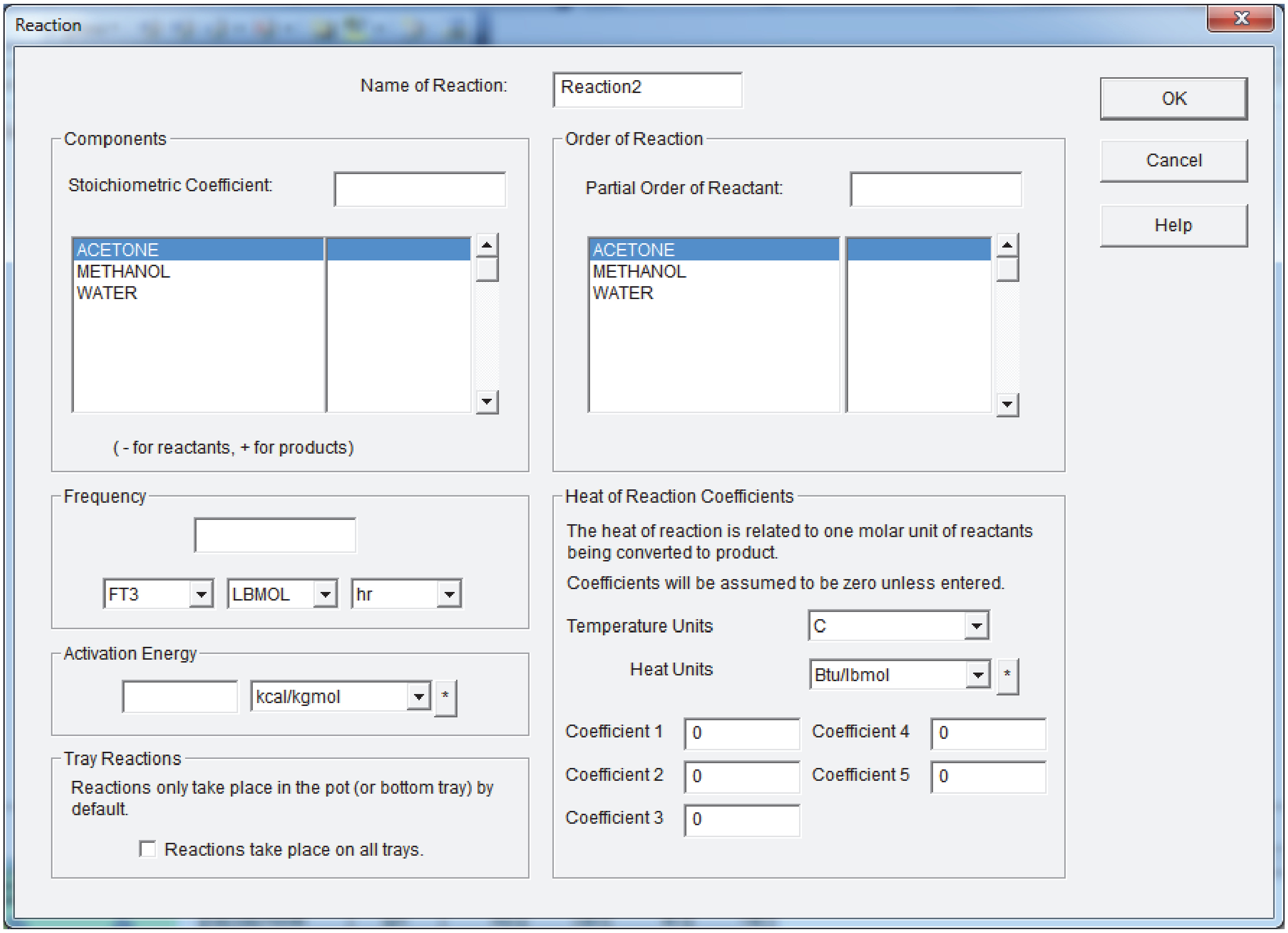


Figure 4: Reaction Dialog (from batch1.psd)

Data Item	Description
Name of Reaction	Enter a name for the reaction.
Stoichiometric Coefficient	Select a component from the list by clicking on it and entering a coefficient value.
Partial Order of Reactant	Select a reactant from the list by clicking on it and enter the number. Reactions are assumed to follow power law kinetics.
Frequency	Reactions are also assumed to have an Arrhenius type expression for the rate constant. Hence, the reaction rate constant can be written. K = f * exp(-E/RT) Make sure that the units of the frequency factor are consistent with the units of the rate expression. Enter a frequency factor; then select the units from the three drop-down lists to complete the frequency expression.
Activation	Enter a value to use as an activation energy for a reaction.
Reactions take place on all trays	Reactions only take place in the pot (or bottom tray) by default. Click this checkbox to allow reactions to take place on every tray instead of just the bottom one.
Temperature Units	Open a drop-down list and select the temperature units to use.
Heat Units	Open a drop-down list and select the heating units to use.
Coefficients 1-5	Coefficients are assumed to be zero unless you enter a value. Enter a coefficient value.

Heat of reaction data must be explicitly supplied in the form of a polynomial in temperature. The heat of reaction is related to one molar unit of reactants being converted to product.

Optional Specifications

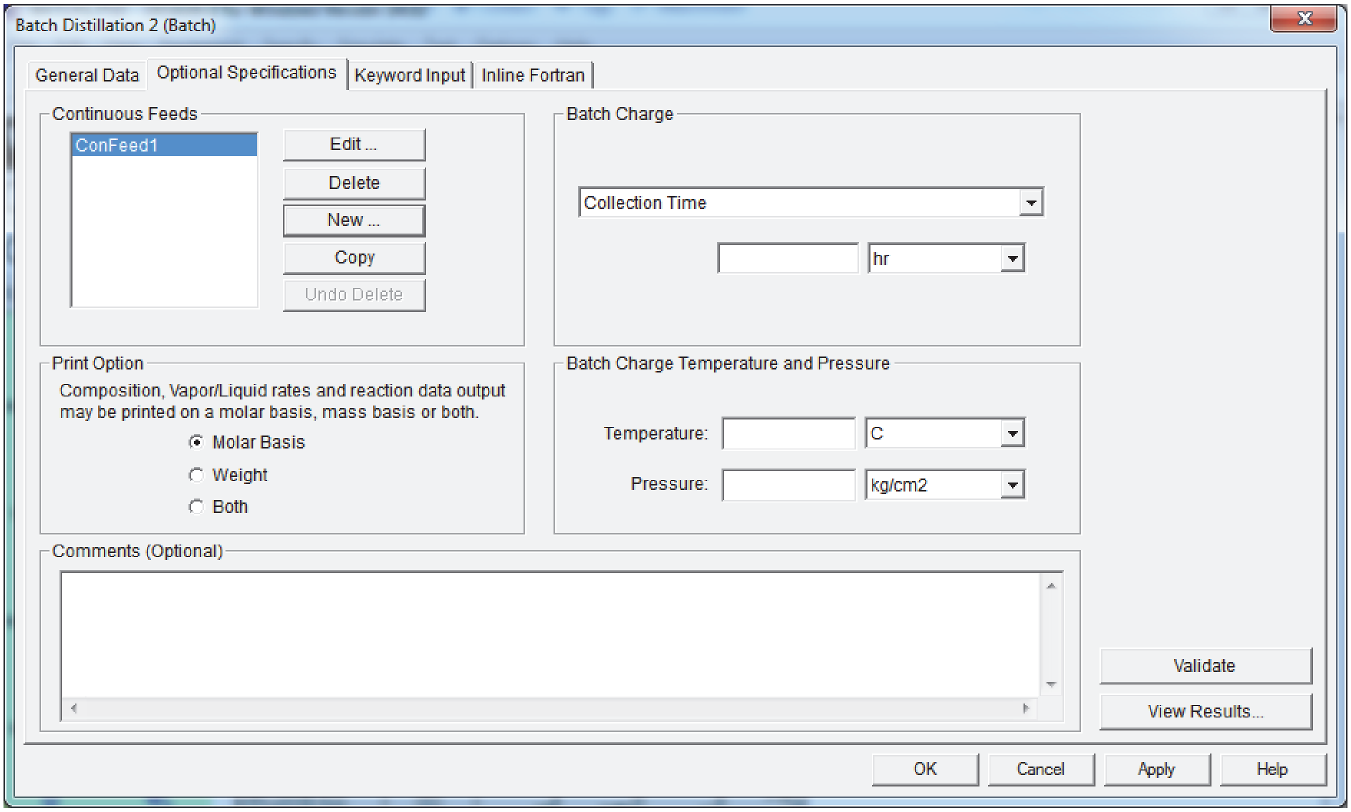


Figure 5: Optional Specifications (from batch1.psd)

Data Item	Description
Continuous Feeds	Click the New button to display the Batch Distillation Continuous Feeds dialog that you can use to create a new feed. Select an existing continuous feed and click the Edit button to change the feed's parameters. Select an existing continuous feed and click the Delete button to remove that feed. Select an existing continuous feed and click the Copy button to replicate the feed. You can then use the Edit function to make any minor changes to the copied feed.
Batch Charge	Select either 1) Collection Time; then enter a time period for which the batch feed stream is to be charged to the still, and select a unit; 2)Total Charge; then enter the total quantity of the batch charge and select a unit.
Print Option	Composition, Vapor/Liquid rates and reaction data output may be printed on a molar basis, mass basis, or both. Select either: Molar Basis, Weight, or Both.
Batch Charge Temperature and Pressure	If you do not enter values for these fields, the batch feed temperature and pressure will be determined by the batch feed stream.
Temperature	Enter a temperature and select a unit.
Pressure	Enter a pressure and select a unit.

Continuous Feeds

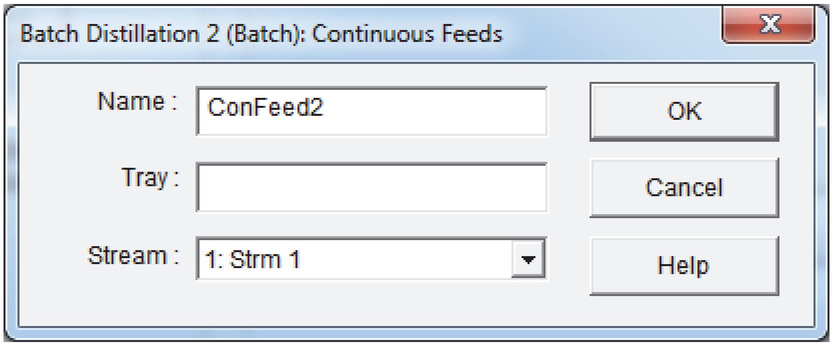


Figure 6: Continuous Feeds Dialog (from batch1.psd)

Data Item	Description
Continuous Feeds	Enter a name for the feed.
Tray	Enter a tray number.
Stream	Open the drop down list and select the stream to use for the feed.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. In future releases of the product, these instructions will be replaced with fill-in-the-blank dialog boxes. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this dialog. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide. Use **the Inline Fortran** tab to enter this information.

Batch Examples

There are several sample flowsheets in “Chapter 11: Batch Distillation Column Samples - c:\designii\samples\equipmnt\batch” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.5: Component Splitter

General

The Component Splitter sends an amount of each component in the inlet stream, which you specify, to the top outlet stream. It is generally used for applications involving individual component separation and may be used for some shortcut column calculations.

Details

The Component Splitter sends a specified fraction of each component in the feed, or inlet stream, to the first outlet stream (top). The rest of the feed is put in the second outlet stream (bottom). The temperatures of the outlet streams can be set to the feed temperature, bubble point, dew point, or a specified temperature.

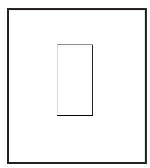
Up to six inlet streams may be connected to this module.   Multiple inlet streams will be mixed by the Component Splitter. Two outlet streams must be coded. This module can be used to decant water, remove H2S and CO2 for shortcut column calculations, and for many other applications involving individual component separation.

Shortcut column calculations can be done with the Component Splitter with more flexibility than the Shortcut Fractionator module. The fractionator module is useful if you have both a condenser and a reboiler and only a top and bottom product. For other columns, the Component Splitter can be used. This allows you to obtain better guesses for the top and bottom product temperature in rigorous distillation. Also, the total reboiler requirements can be obtained by performing a heat balance around the column.

Please see the online **DESIGN II** Help, topic ***Equipment/Component Splitter*** or the **DESIGN II** Unit ModuleReference Guide ***Chapter 6: Component Splitter*** for more details.

Symbols

The Component Splitter unit module has one symbol:



The Component Splitter module requires that one inlet and two outlet streams be connected to the module.   Up to six inlet streams may be connected to the module.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

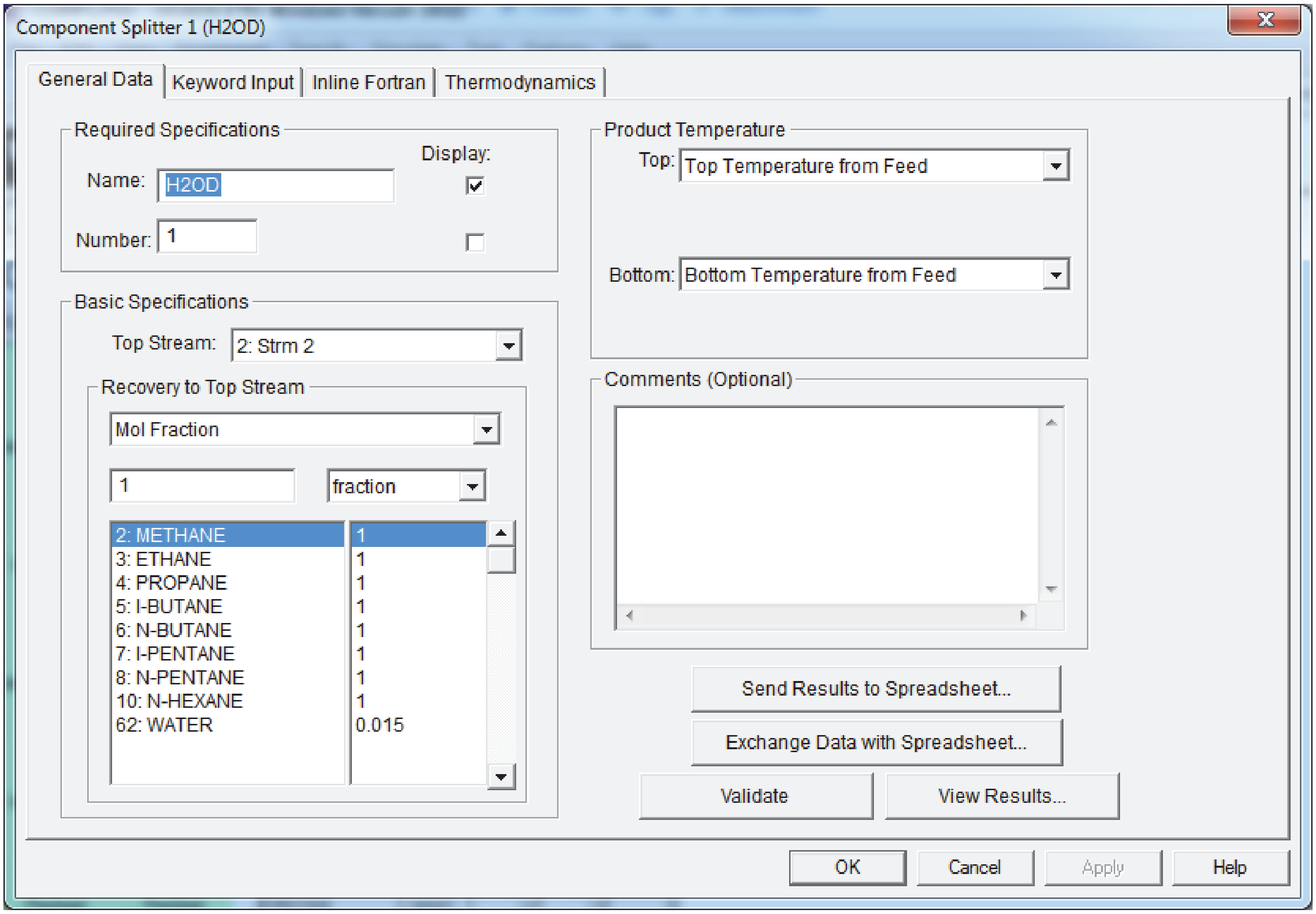


Figure 1: Component Splitter General Data Dialog (from comp1.psd)

General Data

This dialog is used to enter the basic specifications for the Component Splitter Module.

Component recoveries entered on this dialog require that a component list be specified for the simulation with the Components tab of the **Thermodynamic and Transport Methods** dialog (**Specify** menu/**Components**). If these components have been specified; then they will appear in the component list for this dialog.

Data Item	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Top Stream	Select the top stream from the list of all streams attached to the component splitter.
Recovery to Top Stream	This allows you to enter the flowrate or fraction of each component that goes to the first output stream. Select one of the following: Mol Fraction, specifies that top recoveries are entered as fractions of the components in the inlet stream. Comp Molar Flow, specifies that top recoveries are entered as molar flows of the components in the inlet stream. Comp Mass Flow, specifies that top recoveries are entered as mass flows of the components in the inlet stream. Component Vapor Volume Flow, specifies that top recoveries are entered for the components in the inlet stream in units of volume for a gas. Component Liquid Volume Flow, specifies that top recoveries are entered for the components

in the inlet stream in units of volume for a liquid.

After making your selection, select a component in the list. Then, enter a value or change an existing value and select a unit.

The Product Temperature option allows you to set the temperature of the top and bottom products for the component splitter. You can enter the temperatures directly or use bubble and dew point specifications. If none of these commands are used, the temperatures of the top and bottom outlet streams will be set to the temperature of the inlet stream.

Top — Select one of the following:
- Top Temperature from Feed: Sets the top temperature to the temperature of the feed.
- Top Temperature Out: Allows you to enter the top temperature. Enter a value and select a unit.
- Top Bubble Temperature: Sets the top temperature to the bubble point temperature of the top product.
- Top Dew Temperature: Sets the top temperature to the dew point temperature of the top product.

Bottom — Select one of the following:
- Bottom Temperature from Feed: Sets the bottom temperature to the temperature of the feed.
- Bottom Temperature Out: Allows you to enter the bottom temperature. Enter a value and select a unit.
- Bottom Bubble Temperature: Sets the bottom temperature to the bubble point temperature of the bottom product.
- Bottom Dew Temperature: Sets the bottom temperature to the dew point temperature of the bottom product.

Comments — Enter any optional notes about the component splitter.

Optional Specifications: Keyword Input

Use the **Keyword Input** tab to enter this information. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this dialog. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II General Reference Guide. Use the Inline Fortran tab to enter this information.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Component Splitter Examples

There are several sample flowsheets in “Chapter 13: Component Splitter Samples - c:\designii\samples\equipmnt\comspl” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.6: Compressor

General

The Compressor is used to compress a vapor to a specified outlet pressure or to a pressure limited by a specified work available. The only difference between the compressor types is the symbol used.

Details

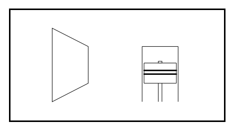
The Compressor compresses a vapor to a specified outlet pressure or to a pressure limited by a specified work available. Up to five inlet streams can be connected to the module. They will be adiabatically mixed to the lowest feed pressure before compression. One or two outlet streams are allowed for this module.

When liquid is found in the suction of the compressor, a warning message is printed in the DESIGN II output file. If the Entropy calculation is specified, a rigorous mixed-phase calculation is performed. If Entropy calculation is not specified, the liquid is compressed as if it were a vapor. If the vapor fraction is less than 0.9, the module shifts to a pump calculation.

Please see the online **DESIGN II** Help topic ***Equipment/Compressor*** or the **DESIGN II Unit Module** Reference Guide ***Chapter 7: Compressor*** for more details.

Symbols

The Compressor unit module has two symbols:



The Compressor module requires that one inlet and one outlet stream be connected to the module. Up to five inlet streams may be connected to the module. If you connect two outlet streams to the compressor, the top stream will contain vapor and the lower stream will contain any liquid. You can also use the Flash module for phase separations of mixtures in the outlet stream.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

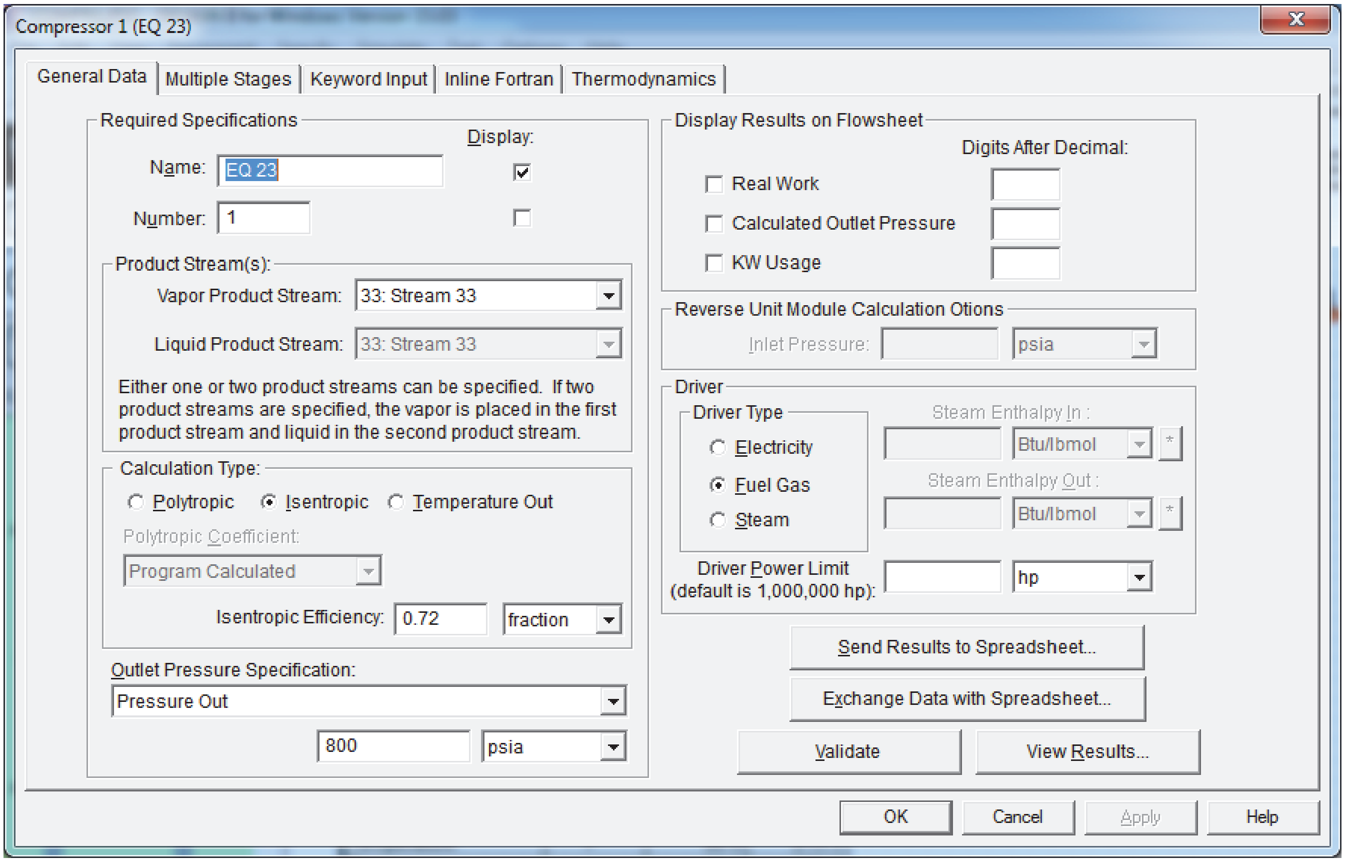


Figure 1: Compressor Dialog (from compr1.psd)

General Data

This tab is used to enter the basic specifications for the Compressor module.

Data Item	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Product Stream	You can select a Vapor Product Stream and Liquid Product Stream; either one or two product streams can be specified. If two product streams are specified, the vapor is placed in the first product stream and the liquid in the second product stream.
Calculation Type	Select either: Polytropic: The compressor will be modeled following a polytropic path, using the polytropic coefficient, between the inlet and outlet conditions. If you select this choice, you can then select Program Calculated (enter the Polytropic Efficiency) or Polytropic Coefficient (enter a Polytropic Coefficient and Polytropic Efficienty), along with an Outlet Pressure Specification. Isentropic: The compressor will be modeled following an isentropic path between the inlet and outlet conditions. You can enter an Isentropic Efficiency along with an Outlet Pressure Specification.
Outlet Pressure Specification	Select one of the following: Pressure Out: The discharge pressure of the compressor. Delta Pressure: Enter the desired pressure change or delta pressure. The default is twice the suction pressure.

Dew Point Temperature: The dew point pressure at the specified temperature is calculated ; then used as the discharge pressure for the compressor calculation.

Bubble Point Temperature: The bubble point pressure at the specified temperature is calculated ; then used as the discharge pressure for the compressor calculation.

Once you have made your selection, enter a value and select a unit. If you selected Pressure Out for the Outlet Pressure Specification: You can enter multistage compressor specifications.

Display Results on Flowsheet: You can optionally display the following results on the flowsheet: Real Work, Calculated Output Pressure, and KW Usage. If you select to display one or more of these results, enter the number of digits to display to the right of the decimal.

Driver Type: Select one of the following:

Electricity: Specifies that the compressor is driven electrically. DESIGN II will report the kilowatts required for the electric motor.

Fuel Gas: Specifies that the driver for the compressor is based on fuel gas. The amount of fuel gas consumed assumes 980 BTU/SCF (8.639 E+05 KJ/KGMOL) and 8000 BTU/HR (3.14 E-03 KJ/SEC/WA).

Steam: Specifies that the compressor is steam driven. If you choose this driver, you need to enter the steam enthalpy at its inlet and outlet.

Steam Enthalpy In: For the Steam driver type, enter the enthalpy of the steam at the inlet of the driver. Enter a value and select a unit.

Steam Enthalpy Out: For the Steam driver type, enter the enthalpy of the steam at the outlet of the driver. Enter a value and select a unit.

Driver Power Limit: For all driver types, enter the available work capacity. The available work may limit the discharge pressure if the required horsepower to reach the specified pressure exceeds this value. Enter a value and select a unit.

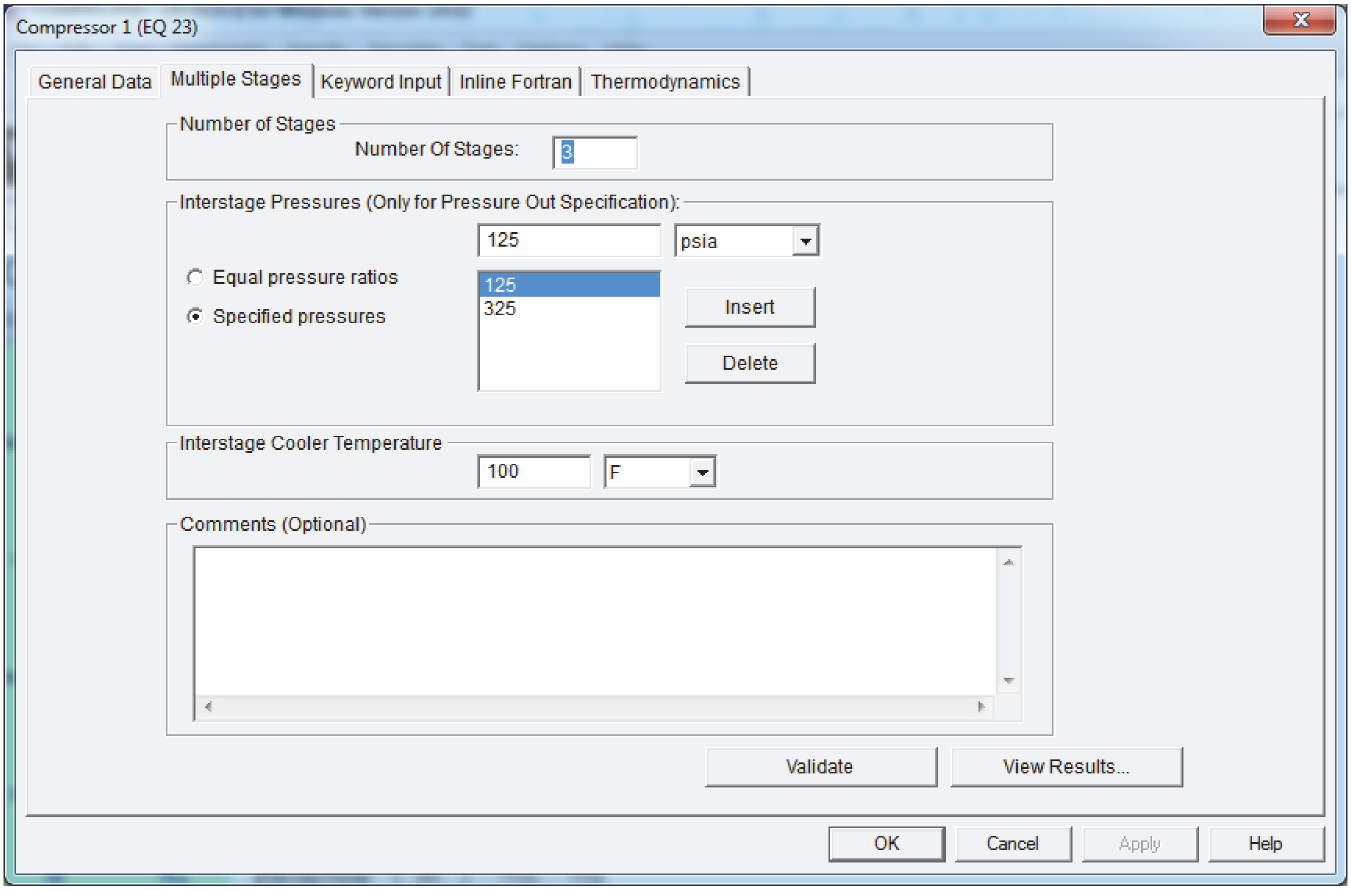


Figure 2: Multiple Stages Tab (from compr1.psd)

Multistage

Use this tab to enter specifications for multistage compressors. The specifications include the number of stages, interstage discharge pressures and intercooler discharge temperature. Multistage compressors require that the Pressure Out specification be set on the compressor basic specifications dialog.

If you are specifying the interstage discharge pressures then the number of pressures entered must be one less than the number of stages. The compressor final discharge pressure is entered in the compressor basic specifications dialog.

The compressor interstage pressures must be in ascending order and non-blank. If you do not choose to enter the interstage pressures, DESIGN II will perform calculations assuming equal compression ratios for each stage.

The maximum number of stages in a compressor is 10. The compressor interstage coolers are only used for the interstage discharges. Final stage discharge will not be cooled.

If you do not enter a value for the intercooler temperature, DESIGN II will use the compressor feed temperature as a default value.

Optional Specifications: Keyword Input

Use the **Keyword Input** tab to enter this information. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this dialog. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide. Use the **Inline Fortran** tab to enter this information.

Optional Specifications: Thermodynamics

This dialog provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Compressor Examples

There are several sample flowsheets in “Chapter 12: Compressor Samples - c:\designii\samples\equipmnt\compre” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.7: Continuous Stirred Tank Reactor

General

The CSTR model assumes a steady state flow of species for the reactor and that the concentrations of species and temperature are uniform throughout the reactor. It uses a Quasi-Newton method to solve a series of equations which determine the outlet compositions for the reactor. Multiple reactions can be specified for the CSTR. Mass and energy balance calculations assume perfect mixing conditions. In addition, you can employ the default power law kinetics model or user specified models for particular reactions.

Details

There are four CSTR types that you can model using DESIGN II:

- Isothermal
- Temperature out
- Adiabatic and Adiabatic with a specified duty
- Results

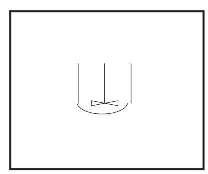
All the important parameters, such as reaction expressions, calculated duty, reactor type, and other related specifications will be reported in the Equipment Summary section of your output. The compositions, temperature, and pressure of the CSTR product will be reported in the Stream Summary and Detailed Stream sections.

Please see the online **DESIGN II** Help topic ***Equipment/Continuous Stirred Tank Reactor*** or the **DESIGN II Unit Module** Reference Guide ***Chapter 9: Continuous Stirred Tank Reactor*** for more details.

Symbols

The Continuous Stirred Tank Reactor module has one symbol:

The CSTR module requires that one inlet stream and one outlet stream be connected to the module.



Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

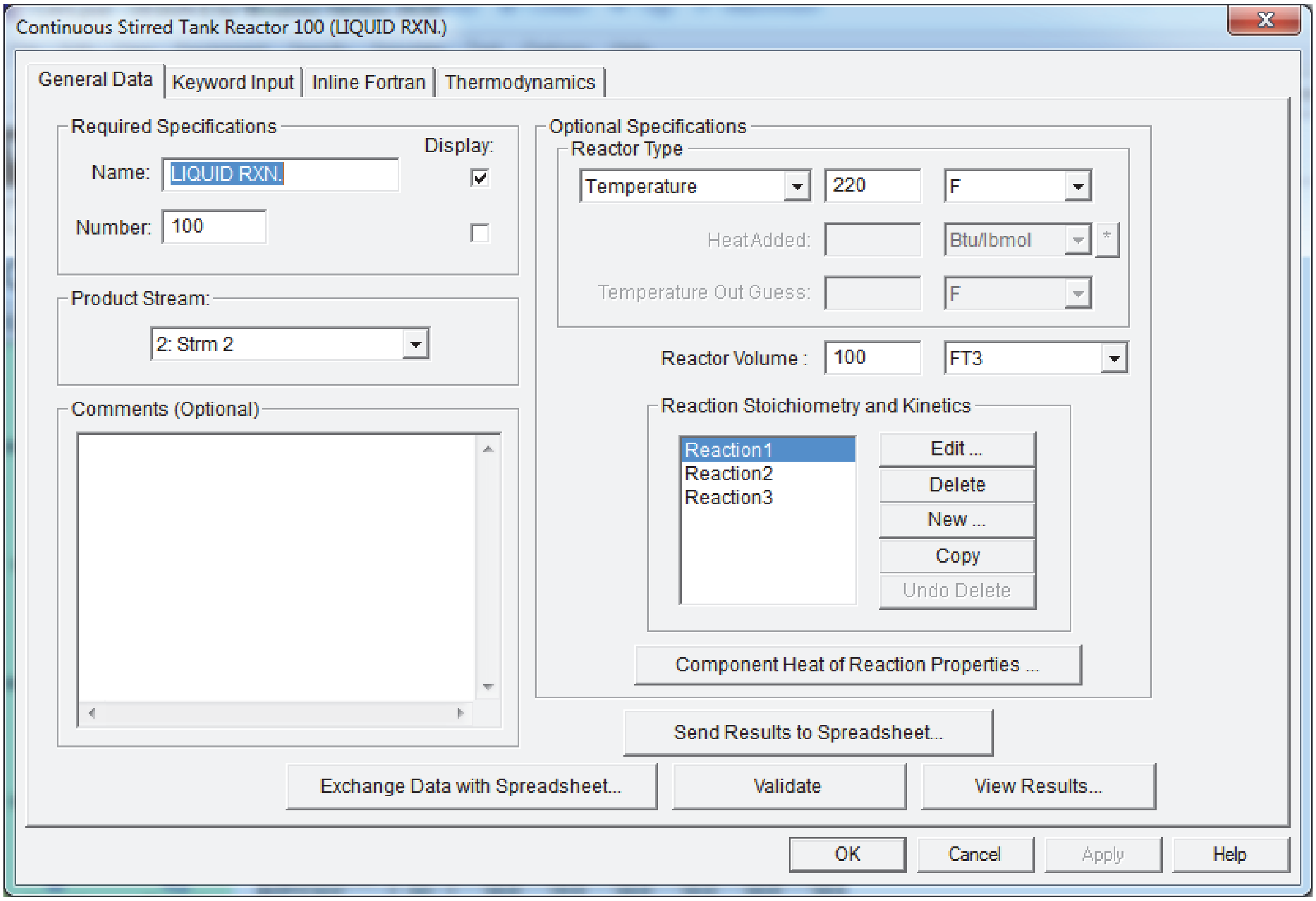


Figure 1: Continuous Stirred Tank Reactor Dialog (from cstr1.psd)

General Data

This dialog is used to enter the basic specifications for the CSTR module.

Data Item	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Product Stream	Select a stream to use as the product stream.
Reactor Type	Select one of the following: Isothermal Reactor, where the product stream enthalpy is calculated using the feed stream temperature; this is the default type. Temperature Adiabatic Reactor, where the product stream enthalpy is equal to the feed stream enthalpy plus the heat of reaction, and the temperature of reactor product is calculated. If you selected Temperature for reactor type, enter the required outlet temperature and select a unit. If you selected Adiabatic Reactor: Heat Added: enter an optional heat duty specification value and select a unit. To enter base enthalpy and quantity units, click the *.

	Temperature Out Guess: enter an initial guess for the outlet temperature value and select a unit.
Reactor Volume	Enter the volume of the reactor.
Reaction Stoichiometry and Kinetics	You can create a new set of stoichiometry and kinetic parameters for the reaction. You can also edit, delete or copy a set. To create a new set of reaction parameters, click the New... button. The Reactions dialog displays. To edit an existing set, click on the name in the list then click the Edit button. To remove a set, click on its name in the list then click the Delete button; a message asks you to confirm the deletion. To duplicate a set, click on its name in the list then click the Copy button.
Component Heat of Reaction	Click this button to display the Component Heat of Reaction Properties tab on the Component/Thermodynamic Methods dialog.

Reaction Stoichiometry and Kinetics

This tab is used to enter the reaction specifications for the CSTR module.

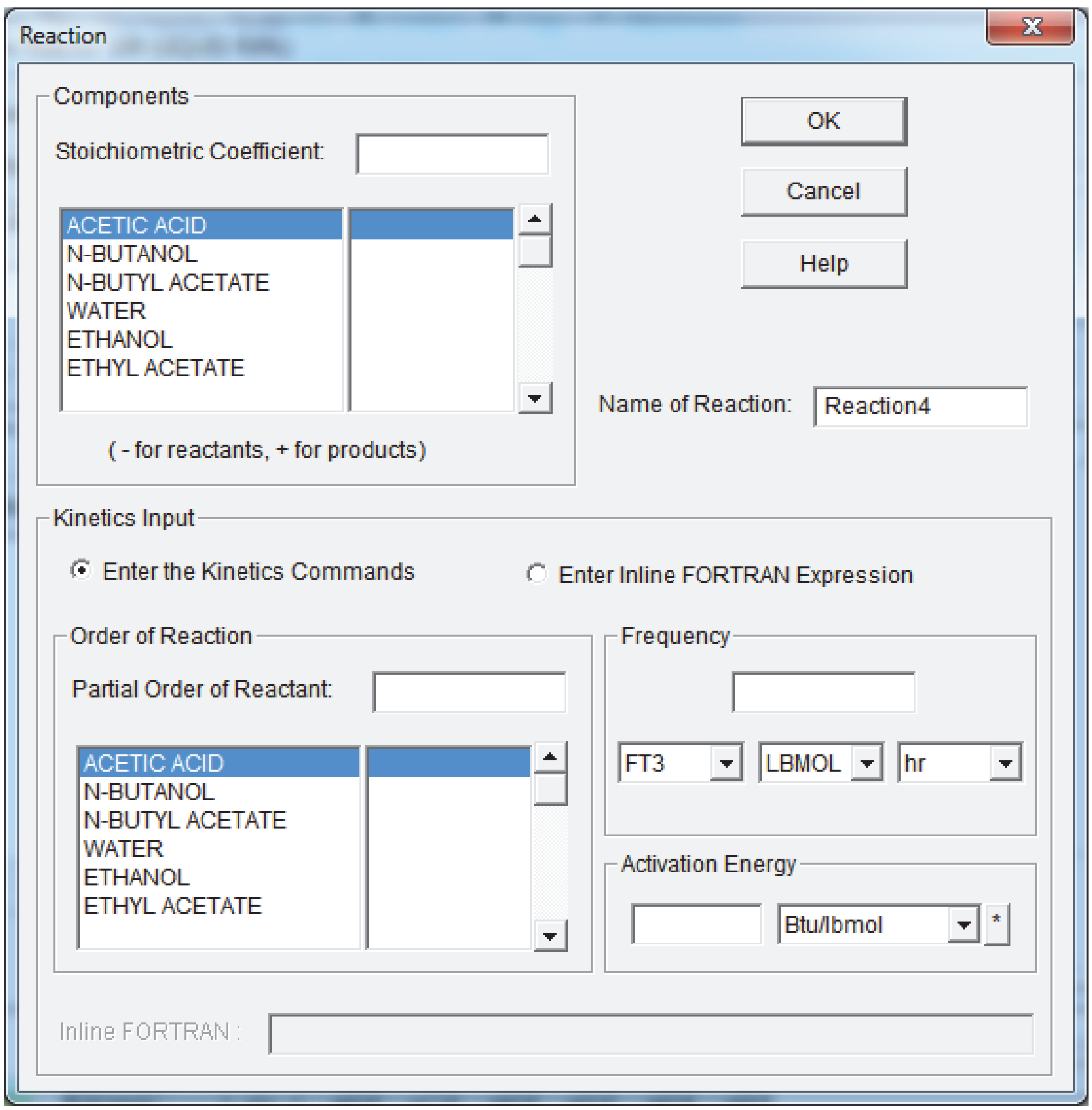


Figure 2: Reaction Dialog (from cstr1.psd)

Data Item	Description
Stoichiometric Coefficient	Select a component from the list for which you want to enter a stoichiometric coefficient by clicking on it. Enter a stoichiometric coefficient for the selected component. Use a negative value for reactants and a positive value for products.
Name of Reaction	View a name for the reaction, or enter a new one.
Kinetics Input	Select either:

Enter the Kinetics command

Enter Inline FORTRAN expression

	If you selected Enter the Kinetics command, you can enter input for the Order of Reaction, Frequency, and Activation Energy sections.
	If you selected Enter Inline FORTRAN Expression, you will create an Inline FORTRAN statement.
Partial Order of Reactant	Select a component from the list by clicking on it; then enter a partial order of reactant.
Frequency	Enter a frequency factor and select the unit specifications to complete the expression.
Activation Energy	Enter an activation energy and select a unit. To change the base enthalpy and quantity units, click here *.
Inline FORTRAN	If you selected Enter the FORTRAN expression, type the FORTRAN code to use as the kinetics input.

Component Heat of Reaction Properties

You can use this tab on the **Component Methods** dialog to enter heat of reaction properties for any components between 100-9999. You can specify a Heat of Formation or a Heat of Formation (Liquid) but not both.

This dialog can be accessed via the Specify/ Components menu item.

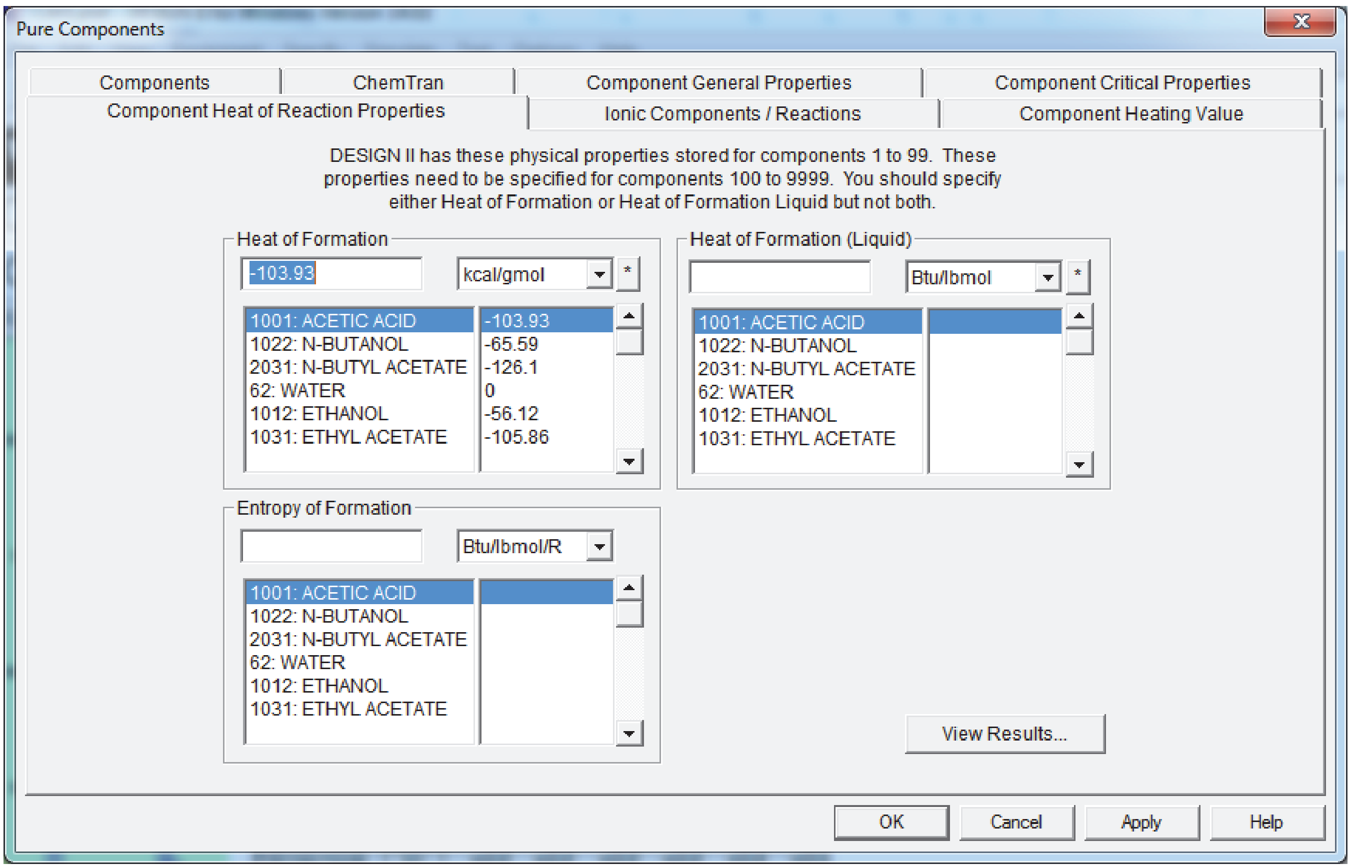


Figure 3: Component Heat of Reaction Properties tab on Components Methods Dialog (from cstr1.psd)

The lost work calculation (thermodynamic efficiency of a process) requires heat of formation and entropy of formation for each component in the process when a reactor module (REA, EQU REA, PLU, REA, CSTR, etc.) is used. In addition, for calculations which include any REActor module, both the heat of formation and entropy of formation for all components in the system must be known before its calculations can be performed.

Heats of formation and entropies of formation for component ID numbers 1 through 99 are automatically available from the Pure Component Database. If you selected any components 100-9999 (by entering components on this dialog's Components tab), you must set a Heat of Formation OR a Heat of Formation (Liquid) for each one. Temperature basis for heat and entropy of formation values is 25 C.

Select a component; then enter the parameter value using the edit box and select a unit.

Heat of Formation	Select a component by clicking on its name in the list. Enter the heat of formation for the component. You can select a unit.
OR	
Heat of Formation (Liquid)	Enter the heat of formation on a liquid basis for the component. You can select a unit. To change the base enthalpy and quantity units, click here *.
Entropy of Formation	You can use this with either Heat of Formation or Heat of Formation (Liquid). Select a component by clicking on its name in the list. Enter the entropy of formation for the component. You can select a unit.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this dialog. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide. You can use the **Inline Fortran** tab to enter this data.

Optional Specifications: Thermodynamics

This dialog provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II Reference Guide.

CSTR Examples

There is a sample flowsheet in “Chapter 15: Continuous Stirred Tank Reactor (CSTR) Samples - c:\designii\samples\equipmnt\cstr” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.8: Controller

General

The Controller is used to set or vary a specific condition within the flowsheet. Like all equipment modules, the Controller is located in the flowsheet by means of input and output stream numbers. However, the Controller does not alter its streams in any way; the stream out is identical to the stream in. The Controller can only measure stream information and cannot change it except via an equipment module.

Details

Passing Information
When passing information, the CONTroller module must be located after the equipment module that calculates the information to be passed, and before the module to which the information is passed. For example, when passing work produced in an EXPander to a COMPREssor, the CONTroller module must be located after the EXPander module. The stream out of the EXPander should be coded into the CONTroller.

Setpoint Convergence
When converging to a setpoint, the CONTroller module should be located after the equipment module that calculates the property being compared to the setpoint. The CONTroller module, when used to obtain a setpoint, changes the order of equipment calculation. After the CONTroller is calculated, calculation order returns to the equipment module where the VARied property is located. Make sure that the equipment that is varied is calculated before the CONTroller. Also you must make sure all other equipment instrumental in calculating the measured property or properties have been calculated. The equipment calculation sequence can be determined by using a CHEck INPut command in the GENeral command section of the input for the first run. If the calculation sequence is not satisfactory, you can change it by using the RECycle SEQuence command in the GENeral section.

Recycle Simulation with CONTrollers
When your simulation involves both a recycle loop and a setpoint convergence CONTroller, certain restrictions apply. A CONTroller loop is formed by the equipment whose specification is varying, the equipment or stream whose property is being measured, and the CONTroller module itself. This CONTroller loop must be either completely inside or completely outside any recycle loop. In particular, the CONTroller loop and the recycle loop must not begin or end with the same equipment in the flowsheet.

A CHEck INPut run should be made first to determine the equipment calculation sequence and recycle streams. If the CONTroller loop is valid, the complete simulation may then be run. For this run, you should omit CHEck INPut, include recycle stream guesses (FLO and TP in GENeral section), and specify a maximum number of recycle loops. You should also consider using the GENeral section command PRInt FREquency = 1. You can set the sequence of equipment calculations using the RECycle SEQuence command.

Changing Stream Parameters
The CONTroller module by itself cannot change stream parameters - it must be used with another equipment module.

Changing Stream Pressure
Use the CONTroller with the PUMp, COMPREssor, EXPander, or VALve module.

Changing Stream Temperature
Use the CONTroller with the HEAt EXChanger or FIRed HEAter.

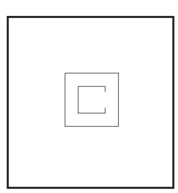
Changing Total Flowrate
Use the CONTroller with the STReam manipulator or the DIVider module.

Please see the online **DESIGN II** Help, topic ***Equipment/Controller*** or the **DESIGN II** Unit Module Reference ***Guide Chapter 8: Controller*** for more details.

Symbols

The Controller unit module has one symbol:

The Controller module requires that one inlet stream and one outlet stream be connected to the module.



Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

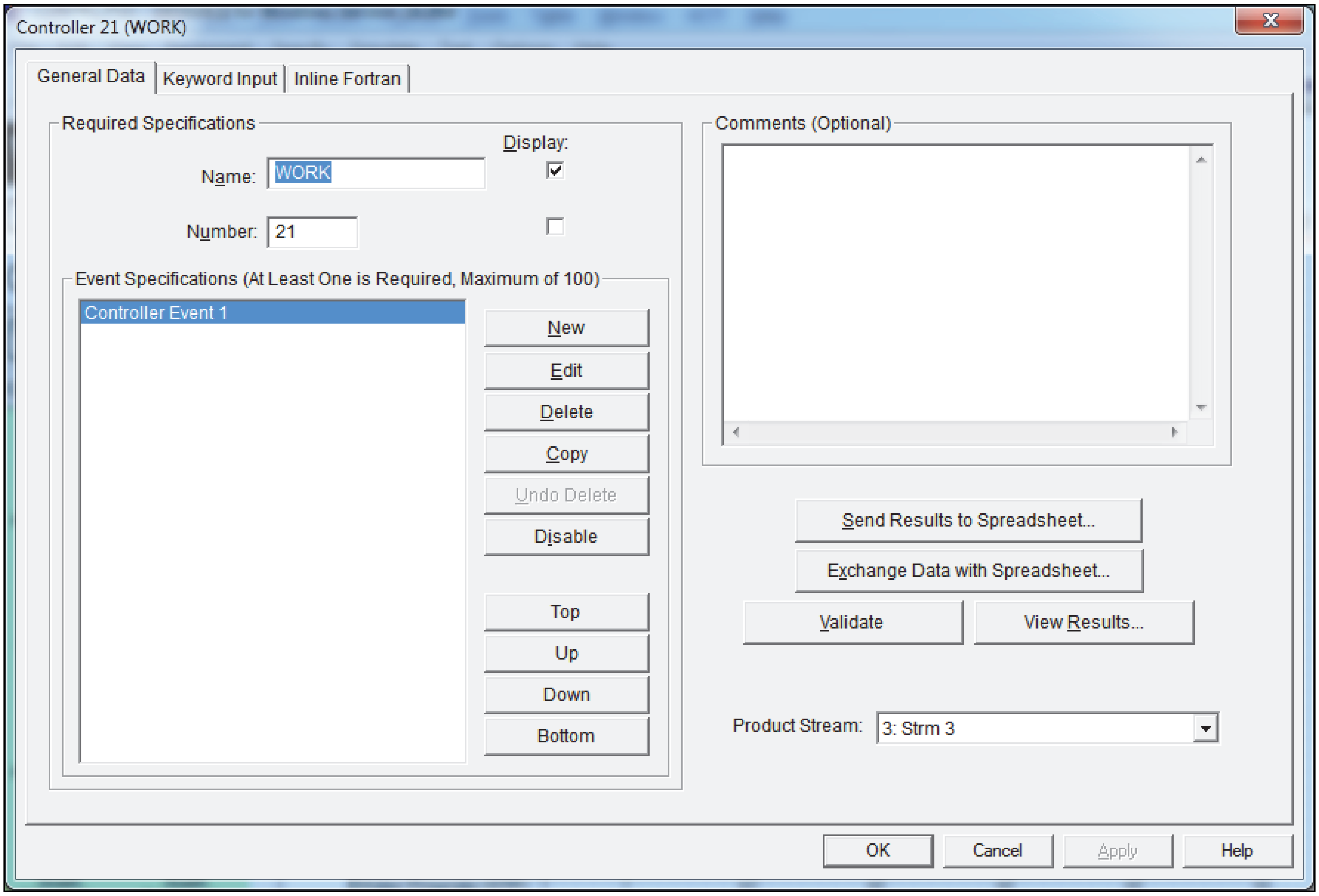


Figure 1a: Controller Dialog (from contr1.psd)

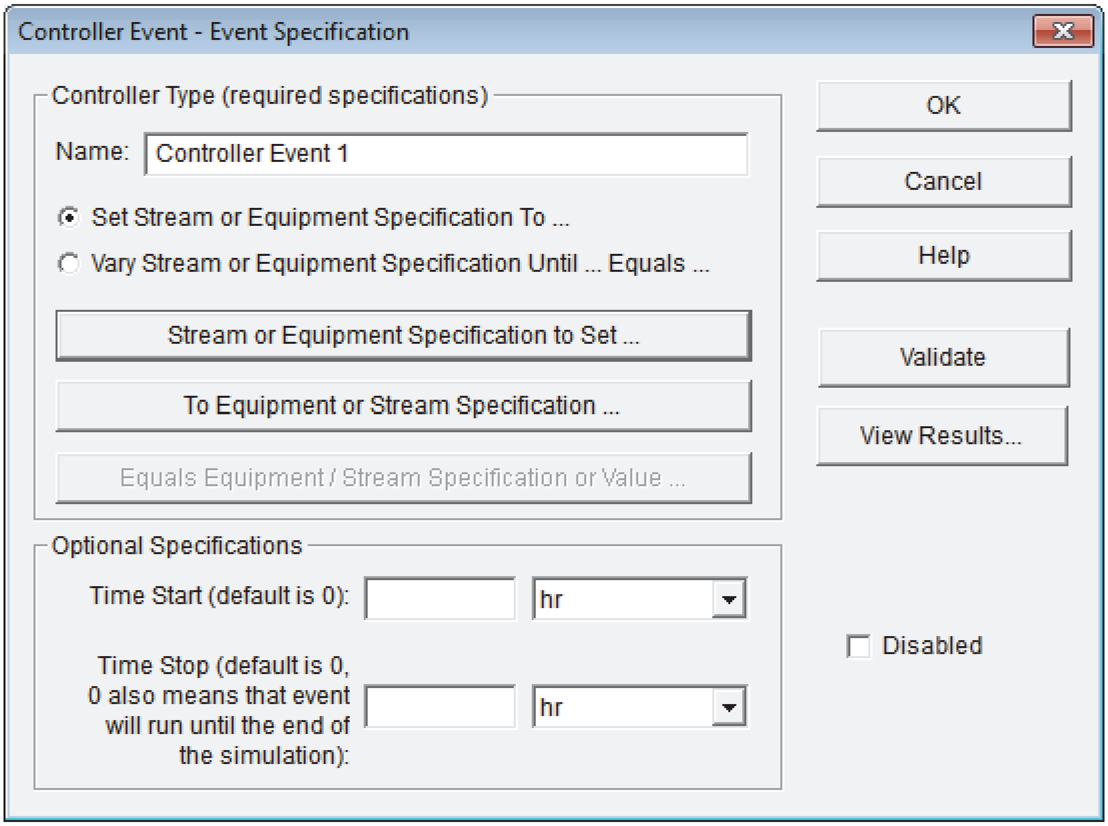


Figure 1b: Controller Dialog (from contr1.psd)

The Set and Vary radio buttons control the contents of the three required Specification dialogs.

Controller Passing Information

SET X1 OF equipment = X2 (SCAle =) FROm ES operator X3 (SCAle=) FROm ES

where:

X1	is an Equipment Module keyword command; e.g. TEM OUT for HEAt EXChanger
equipment	is the Equipment Module identifier and equipment number e.g. HEA EXC 7
X2, X3	are any calculated equipment or stream properties; e.g. CAL DUT for a HEAt EXChanger or FLO COM for a stream. X2 and X3 can also be Equipment Module keyword commands; e.g. TEM OUT for a HEAt EXChanger is a stream as identified by STR and stream number; e.g. STR 18
SCAle =	multiplication factors used to modify X2 and X3 (Default is 1.0).
operator	+(add), -(subtract), *(multiply), /(divide)
NOTE:	operator and X3 are optional

Controller Setpoint Convergence

VARY X1 (MIN =, MAX =, STE =, units) OF equipment UNTIL X2 (SCAle =) FROm ES operator X3 (SCA =) FROm ES= constant (SCA =, BEG =, LOO =, TOL =, units)

OR

X4 (SCA =, BEG =, LOO =, TOL =) OF ES

where:

X1	is an Equipment Module keyword command; e.g. TEM OUT for HEAt EXChanger
equipment	is the Equipment Module identifier and equipment number e.g. HEA EXC 7
X2,X3,X4	are any calculated equipment or stream properties from Tables 1 and 2; e.g. CAL DUT for a HEAt EXChanger or FLO COM for a stream. X2, X3, and X4 can also be Equipment Module keyword commands; e.g. TEM OUT for a HEAt EXChanger constant is a desired setpoint value. Units must be specified if other than American defaults.
ES	is the Equipment Module identifier and equipment number OR the stream number and identifier (e.g. STR 18).

SCAle =	multiplication factors used to modify X2, X3, and X4 (Default is 1.0)
operator	+(add), -(subtract), *(multiply), /(divide)
MINimum value =	is the specified minimum value for X1
MAXimum value =	is the specified maximum value for X1
STEp size =	is the maximum step size to be taken when varying a specification. The initial step will be five percent of the specified step size to determine the slope and direction for the next change.
TOLerance =	is the absolute difference acceptable for setpoint convergence. Default is the relative CONvergence TOLerance specified in the GENeral command section.
BEGin loop =	is the process control loop iteration number at which action of the CONTroller is to begin. Default is first process loop.
LOOps =	Maximum number of controller loops for setpoint convergence. Default is 30.
units	Units must be specified where indicated for MIN =, MAX =, and STE =, and for constant if other than American defaults; e.g. ATM for pressure. See INPUT / OUTPUT UNITS Section for details.
VARY X's	(MIN=, MAX=, STE=, units) of equipment
UNTIL X2	(Scale)FRO name operator X3 (scale) Fro name =
X4	(Scale=, BEGin =, LOOps =, TOL=) OF name

NOTE: X4 can also be constant, X3, operator are optional.

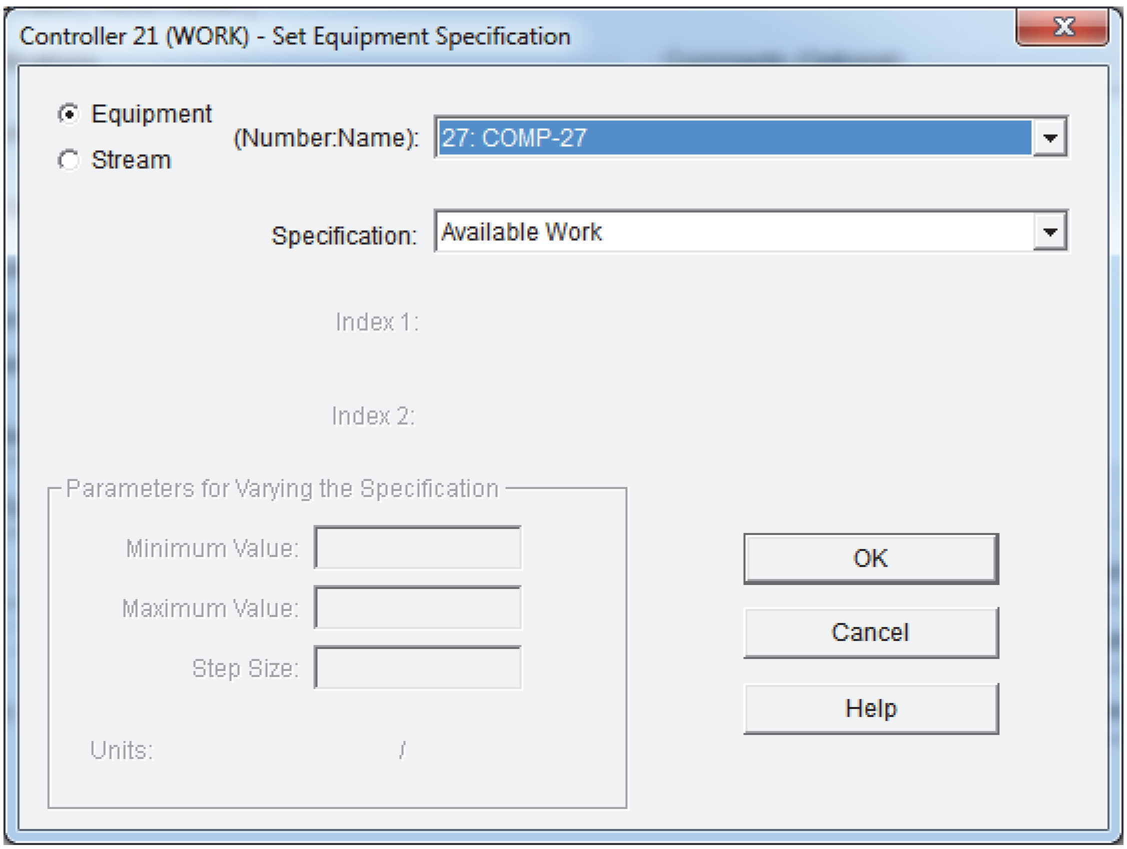


Figure 2: Controller Set / Vary Dialog (from contr1.psd)

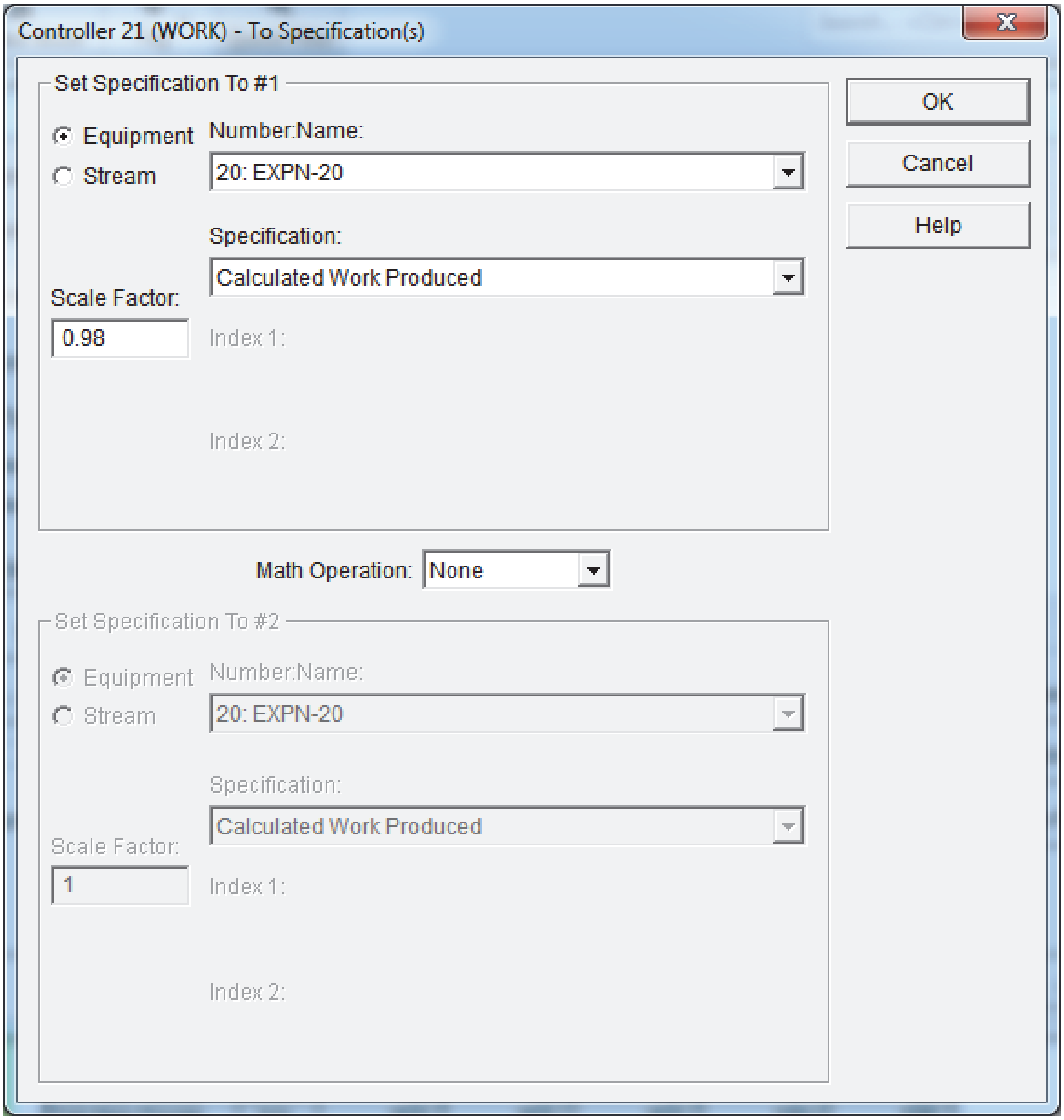


Figure 3: Controller Set To / Vary Until Dialog (from contr1.psd)

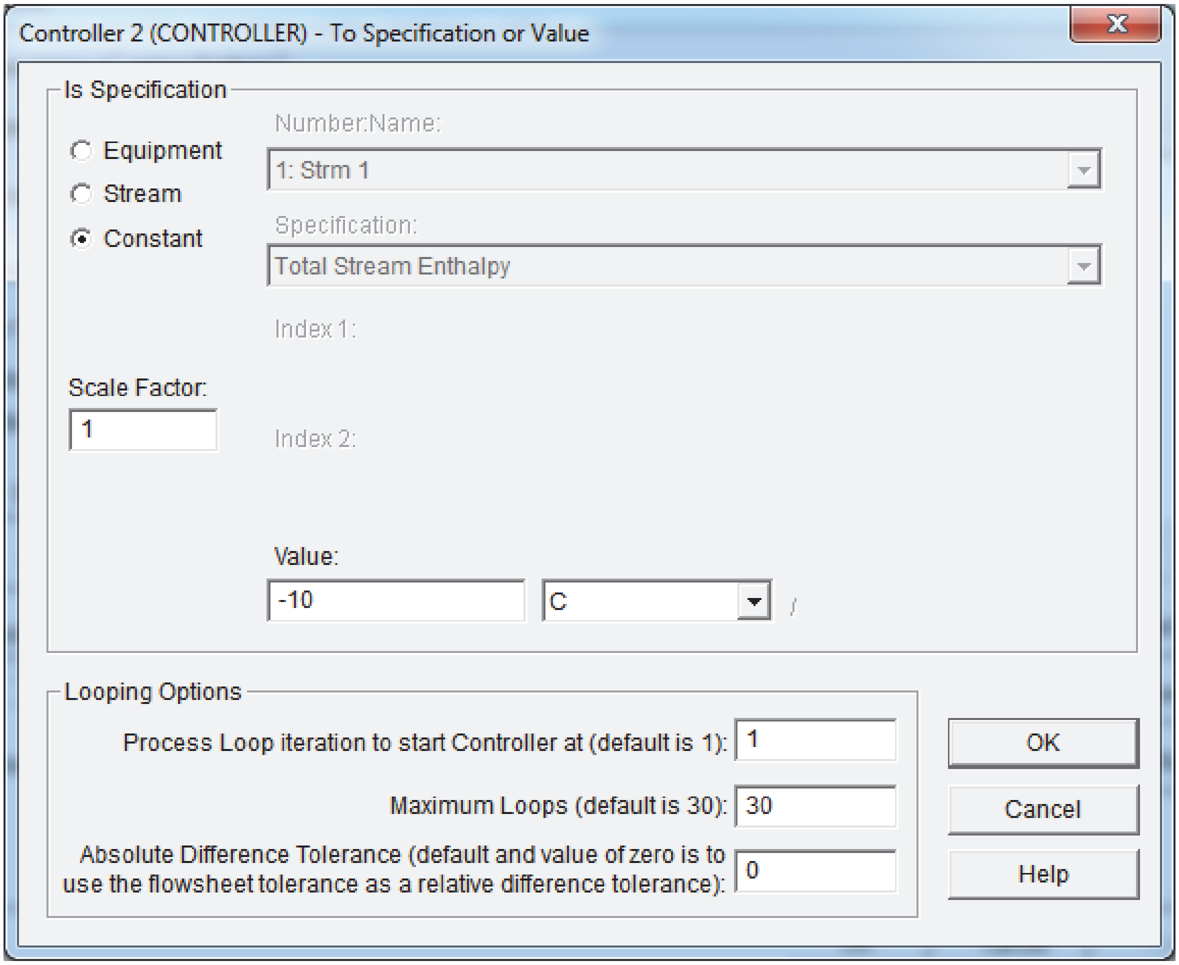


Figure 4: Controller Equals Dialog (from contr4.psd)

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the "C-*" before it and enter the units in parenthesis and any values after the =. Any line beginning with "*" will not be written to the DESIGN II input file. However any line with a "C-" (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this dialog. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide. You can use the **Inline Fortran** tab to enter this data.

Controller Examples

There are several sample flowsheets in "Chapter 14: Controller Samples - c:\designii\samples\equipmnt\cont" of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.9: Depressuring

General

Depressuring systems are used extensively in process plants for protecting vessels from overpressure. Overpressuring has many causes, such as external fire, exothermic reactions, loss of power, inadequate cooling, and blocked lines. Depressuring systems are needed to prevent equipment from rupturing and to provide safety for plant personnel. API RP-520 recommends that vessel pressure should be reduced to 100 PSIA or 50% of design pressure, whichever is lower, in fifteen minutes.

Details

There are four Depressuring types that you can model using DESIGN II:

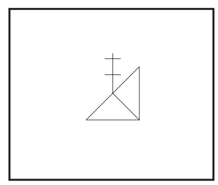
- Isothermal
- Temperature out
- Adiabatic and Adiabatic with a specified duty
- Results

All the important parameters, such as reaction expressions, calculated duty, reactor type, and other related specifications will be reported in the Equipment Summary section of your output. The compositions, temperature, and pressure of the Depressuring product will be reported in the Stream Summary and Detailed Stream sections.

Please see the online **DESIGN II** Help, topic ***Equipment/Depressuring*** or the **DESIGN II Unit Module** Reference Guide ***Chapter 10: Depressuring*** for more details.

Symbols

The Depressuring unit module has one symbol:



The Depressuring module requires that one inlet stream and one outlet stream be connected to the module.

You can connect a second outlet stream to the module so that the maximum flowrate condition will be placed in that stream.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

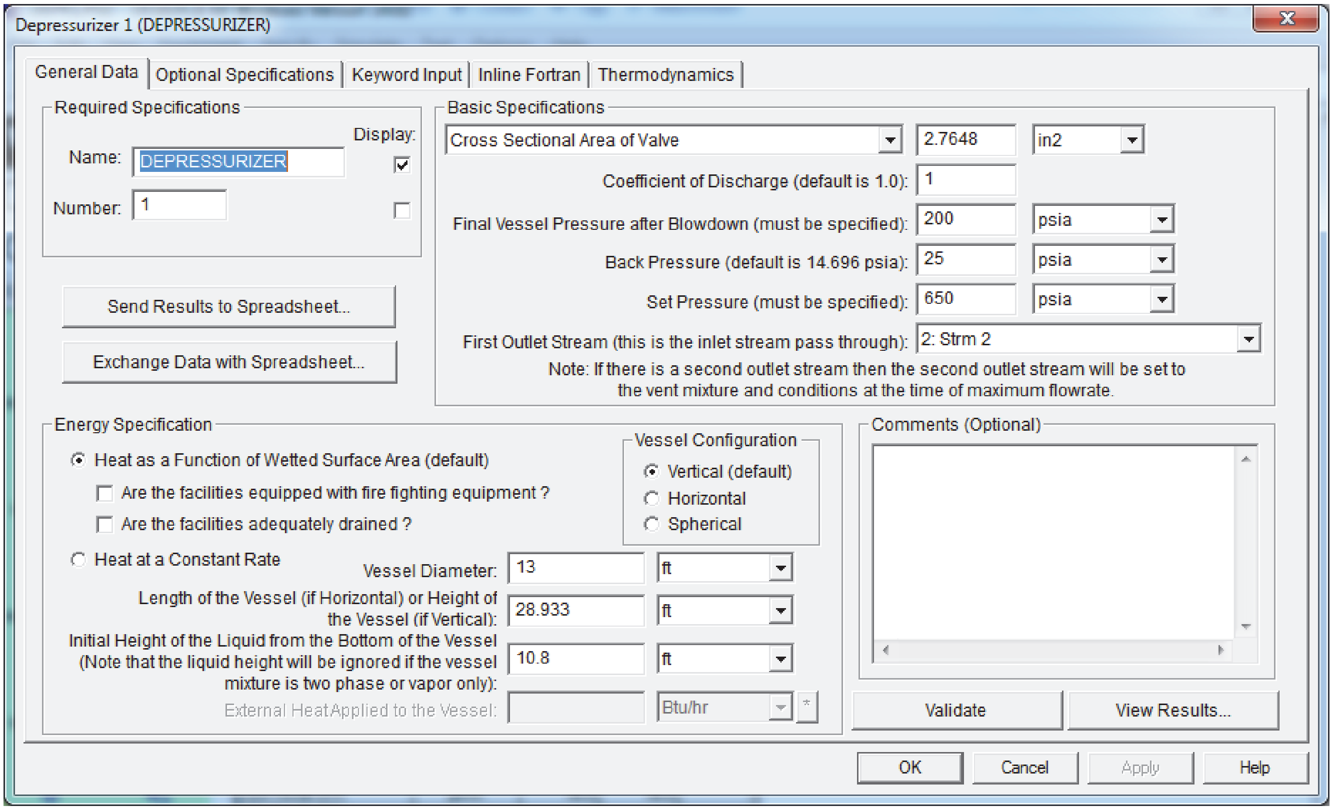


Figure 1: Depressurizer Dialog (from depr1.psd)

The depressuring analysis is based on a vessel which is exposed to an external fire. The program calculates the pressure buildup in the closed vessel until the pressure reaches the set pressure of the valve. The conditions during the blowdown period after the opening of the depressuring valve are also calculated. The blowdown period is divided into several intervals with important variables reported for each time interval.

One inlet and one or two outlet streams may be coded. The second outlet stream is used with PRInt options 2 or 3. This will print stream properties for the vent gas stream at the time of maximum flowrate. The temperature and pressure of the inlet stream are assumed to be the operating conditions of the vessel. A flash is performed to determine the vapor and liquid compositions and initial flow rates. The DEPressuring module will yield more meaningful results if heat as a function of wetted surface area is used. This option includes commands describing the vessel diameter, the vessel orientation, vessel length, and the liquid height.

A calculation will be performed when this input information is not provided, but the vessel size will be based on the vapor and liquid in the feed to this module (lbs/hr are assumed to be total pounds of inventory in the vessel) and a 1 HR residence time. Three assumptions are made for the depressuring analysis:

1. Flow to vessel is stopped (i.e. no fresh feed).
2. The heat involved is provided externally either at a constant rate

   Delta Q = Constant Btu / Time

   or as a function of the wetted surface area.

   Delta Q = 34,500 * F * (Aw ^ 0.82)

   where F is an environmental factor recommended by API RP-520 and Aw is wetted surface area. The wetted surface area factor can be varied to 21000 - only if facilities are equipped with fire fighting equipment and adequate drainage.
3. The vapor flow through the valve is either a choking flow or subsonic flow due to sufficiently large back pressure.

Methods used from API 520 Edition 7

1. All vapor venting for critical and sub critical regimes are included (section 3.6.2 and 3.6.3). Includes corrections for combined rupture disk/valve and only rupture disk options. Correction for balanced bellows valve is available.
2. Steam relief at critical flow (vapor venting option). Also includes rupture disk combinations.
3. Liquid relief for valves not requiring capacity certification. Also includes rupture disk combinations.
4. Two phase (liquid/vapor) relief using Appendix D. of API code. This includes options 1 and 3, critical and sub-critical conditions. Rupture disk combinations are included.

5. Subcooled liquid relief with flashing in orifices and with a non-condensable gas. Rupture disk combinations are included along with corrections for balanced bellows valve. Overpressure correction (Fig D-2) is not included.

General Data

This tab is used to enter the basic specifications for the Depressurizer module.

Data Item	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Cross Sectional Area of Valve	Enter the cross-sectional area for the valve and select a unit.
Coefficient of Discharge	Enter the manufacturer's specification. The default is 1.0.
Final Vessel Pressure after Blowdown	Must be specified. Enter the reduced pressure and select a unit.
Back Pressure	Default is 14.696 psia. Enter a value if a subsonic flow condition is desired (non-choking flowrate). If the back pressure is higher than critical-flow pressure, the resulting vapor flowrate will be lower. Select a unit.
Set Pressure	Must be specified. Enter the pressure at which the valve is to open and select a unit.
First Outlet Stream	If there is a second outlet stream; then the second outlet stream will be set to the vent gas at the time of the maximum flowrate. Select the stream to use as the first outlet stream.
Energy Specification	Select either: Heat as a Function of a Wetted Surface Area Heat at a Constant Rate If you selected Heat as a Function of a Wetted Surface Area: Are the facilities equipped with firefighting equipment: Click the checkbox to indicate yes. Are the facilities adequately drained: Click the checkbox to indicate yes. For either choice you can complete the following: Vessel Diameter: Enter the diameter of the vessel to use for predicting heat transfer based on wetted surface area. Select a unit. Length of Horizontal/Vertical Vessel: Enter the length of the vessel for horizontal or vertical vessel configuration (not applicable for spherical vessel configuration). Select a unit. Height of the Liquid from the Bottom of the Vessel: Enter the height of the liquid from the bottom of the vessel. Select a unit. If you selected Heat at a Constant Rate: External Heat Applied to the Vessel: Enter the constant rate of external heat which will be applied to the vessel. For depressuring without heat, enter 0 and specify a set pressure that is lower than the vessel operating pressure. To enter base enthalpy and time units, click the *. Vessel Configuration: select either vertical, horizontal, or spherical; vertical is the default.

Optional Specifications

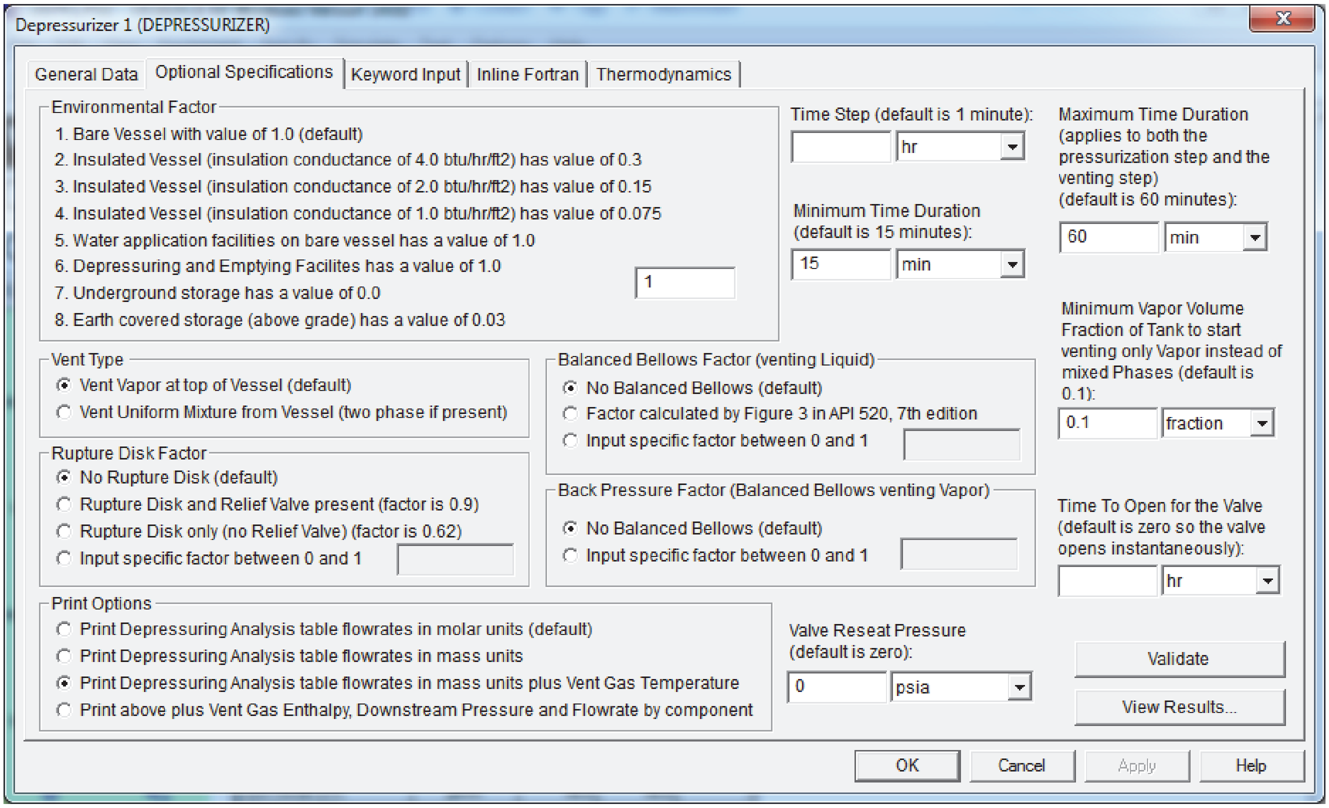


Figure 2: Optional Specifications tab on the Depressurizer Dialog (from depr1.psd)

This tab is used to enter the optional specifications for the Depressurizer module.

Data Item	Description
Environmental Factor	Enter the environment factor in the field, any number 1-8, based on the information presented. The default is 1.
Time Step	Enter the time intervals for the depressuring calculations and select a unit.
Minimum Time Duration	Enter a minimum time required for the depressurizer to complete its specified calculations. The default is 15 minutes, per the Environmental Protection Agency (EPA) requirement in the United States.
Maximum Time Duration	Enter a maximum time duration; this applies to both the pressurization step and the venting step. The default time is 60 minutes.
Vent Type	Select the desired vent type: Vent Vapor at top of Vessel (default) Vent Uniform Mixture from Vessel (two phase if present)
Balanced Bellows Factor (venting liquid)	Select the balanced bellows factor: No Balanced Bellows Factor Calculated by Figure 3 in API 520, 7th edition Input specific factor between 0 and 1: Enter a back pressure correction factor.
Rupture Disk Factor	Select the rupture disk factor: No Rupture Disk Rupture Disk and Relief Valve present (factor is 0.9) Rupture Disk only (no Relief Valve)(factor is 0.62) Input specific factor between 0 and 1: enter a rupture disk factor.
Back Pressure Factor	Select the back pressure factor: No Balanced Bellows Input Specific Factor between 0 and 1: enter a back pressure factor.
Print Options	Select the desired Print Option to use, based on the information presented.
Minimum Vapor Volume	Enter the desired value and select fraction, percent, or PPM. The default value is 0.1.

Fraction of Tank to start venting only Vapor instead of mixed Phases	
Time To Open	The default is zero. Enter the time to open the valve linearly over time when the depressurizer readies itself to open. The amount of flowrate through the valve will be severely cut for a specified time period before it opens to use full throat area. Select a unit. If you do not enter a value, the valve will open to use 100% throat area immediately.
Valve Reseat Pressure	Enter the value and select a unit. The default is zero.

Required Specifications: Keyword Input

Click the **Keyword Input** tab. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Depressuring Analysis Examples

There are several sample flowsheets in “Chapter 16: Depressuring Samples - c:\designii\samples\equipmnt\depres” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.10: Distillation

General

The Distillation module can be used to rigorously model tray-by-tray distillation columns. The column configurations include absorbers, reboiled absorbers, strippers, and fractionators. You may add sidedraws, heaters, coolers, intercoolers, and water decants to these configurations.

Details

The Distillation module is used for rigorous simulation of absorbers, fractionators, strippers and other types of single-column configurations. These configurations are classified into seven major column types:

- Absorber
- Liquid-Liquid Exchanger
- Reboiled Absorber
- Stripper
- Stripper With Total Condenser
- Partial Fractionator
- Total Fractionator

You may add to each of these column types multiple feed streams, product streams, heaters, coolers, and intercoolers. Most single-column units can be modeled, including demethanizers, stabilizers, sour water strippers, amine contactors and regenerators, and azeotropic distillation towers.

There are 18 snap points located on the sides of absorbers for the connection of any combination of feed streams, product streams and sidedraws. This number decreases by one to 17 for columns with a condenser or reboiler, and by two to 16 for columns with both a condenser and a reboiler. A top and bottom product stream must be connected as outlet streams. These connect from the top and bottom of the column if the condenser or reboiler is not present, respectively. They connect from the condenser or reboiler if they are present.

Stream connections to the condenser and the reboiler must reflect the type of model you are using. A vapor outlet from a partial condenser must connect to the top of the separator (reflux accumulator) for the condenser. Liquid from a total condenser, or a sidedraw from the condenser (specify tray 0), must connect to the bottom of the separator. If you specify a kettle type reboiler, the bottom product stream must connect to the symbol representing the reboiler. If you specify a thermosiphon reboiler, the bottom product stream must connect to the snap point at the bottom of the column.

This module can be used for design, modification, or optimization of rigorous distillation columns. Extensive thermodynamic options are available (see Basic Thermo for a summary). With the use of the proper option, the operation of existing units can be duplicated within several percent of actual plant data.

The following topics provide background on the column setup:

Product Specification And Column Control
The flexibility of the DIStillation module also includes the control of the fractionation process. Component purities, recoveries, or ratios can be specified directly in top, bottom, or side products. Product rates can be set in mass, molar, or volumetric units. Reboiler or condenser temperatures and duties can be specified as can reflux ratio or reflux flowrate. Additional specifications can be created with Inline Fortran. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Calculation Techniques
Three methods are available for solving the rigorous tray-to-tray heat and material balances for your column. These are Regular, SUPer, and SUPer PLUs. The trays, partial condensers, and reboilers are modeled as equilibrium stages. Special commands allow the convergence algorithms to be fine tuned to meet user specified tolerances.

Water K-values And Decant
The K-value option you choose for your simulation normally determines whether water (if present) is treated miscibly or immiscibly in the column. "Free" water is not decanted automatically. If you specify a water decant from the condenser or any tray(s), the amount of "free" water formed at the temperature and pressure of the condenser or tray is calculated automatically. You may also specify the solubility of water in the reflux returning to the column. The default solubility will be based on the water-in-kerosene chart from the API Technical Data Book.

Stream Order Convention
Certain conventions are followed when feed and product streams are defined for the Distillation module.

Each feed stream to the Distillation module is assigned a unique number or name. You must assign a stream number on the flowsheet to a feed stream name. This is done with the Feeds dialog. On the DESIGN II keyword command, the feed streams will be listed by DESIGN II for Windows in top-to-bottom order.

Each product stream is assigned a unique number. The product stream numbers are given negative signs and listed in top-to-bottom order on the DESIGN II keyword command generated by DESIGN II for Windows.

Decant streams from the condenser or any tray are not counted as a product and do not require a stream number.

Internal streams, such as reboiler or reflux streams, except when the streams are used in conjunction with an internal stream command are not numbered. Their compositions, flowrates, and other properties are reported automatically in the column results.

Tray Numbering Convention
Certain conventions must be followed when numbering trays in the Distillation module. Tray numbers are used to locate feeds, sidedraws, side heater/coolers, and intercoolers.

All trays are theoretical trays.

Trays are numbered from top-to-bottom. The top tray is always tray one. The number of trays entered does not include a condenser or a reboiler. If present, the program handles them automatically by numbering the condenser tray zero and the reboiler one larger than the number of trays.

Step-by-Step Procedure
Specifying a rigorous Distillation column with DESIGN II is simplified by following a step-by-step procedure:

Define the column type by selecting the combination of condenser and reboiler you want from the Distill 1, Distill 2, Distill 3, or Distill 4 shapes. The shapes will be displayed in the Browser.

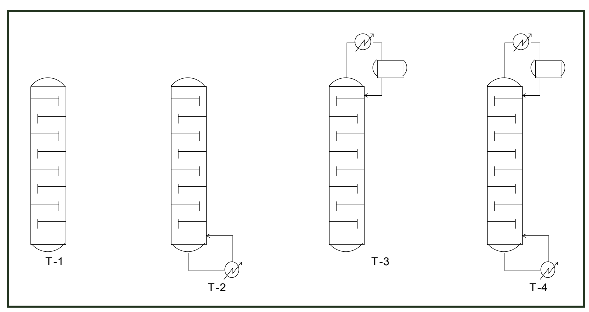
By selecting the dialogs under Required Specs, add any required column configuration specifications. These should include heat and product specifications.

Add any sidedraws, heaters/coolers, intercoolers, and water decants using the Optional Specs dialogs.

With the Keyword dialog under Optional Specs, you add specifications as needed for reboiler or condenser curves, CO2 freeze prediction, extra plots, column sizing, and others.

Please see the online **DESIGN II** Help, topic ***Equipment/Distillation Column*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 11: Distillation Column*** for more details.

Symbols



There are four symbols for the Distillation Column. Distill 1 is used for Absorbers and Liquid – Liquid Exchange Columns. Distill 2 is used for Reboiled Absorber Columns. Distill 3 is used for modeling Strippers (Columns with a partial or toal condenser. Distall 4 is used for modeling columns with both a condenser and a reboiler.

The Distillation module requires that at least one inlet (feed) and two outlet streams (top and bottom products) be connected to the module. The top and bottom outlet streams must be connected to the top and bottom snap points (at the top and bottom of the column), or to the reboiler or condenser if they are present. For thermosiphon reboilers, connect the bottom product to the snap point at the bottom of the column. For kettle reboilers, connect the bottom product to the snap point on the reboiler symbol.

If there is a partial condenser then you must have outlet streams connected to the top and bottom of the condenser. If there is a total condenser then you should only attach an outlet stream to the bottom of the condenser.

The number of sidedraws and inlet streams are limited to the total number of available snap points. In addition to the top and bottom outlet streams, you may connect a total of 9 sidedraws. To determine the number of inlet streams you can connect, subtract the total number of outlet streams from 20 for an absorber; 19 for a column with either a condenser or reboiler; and 18 for a column with both a condenser and a reboiler.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

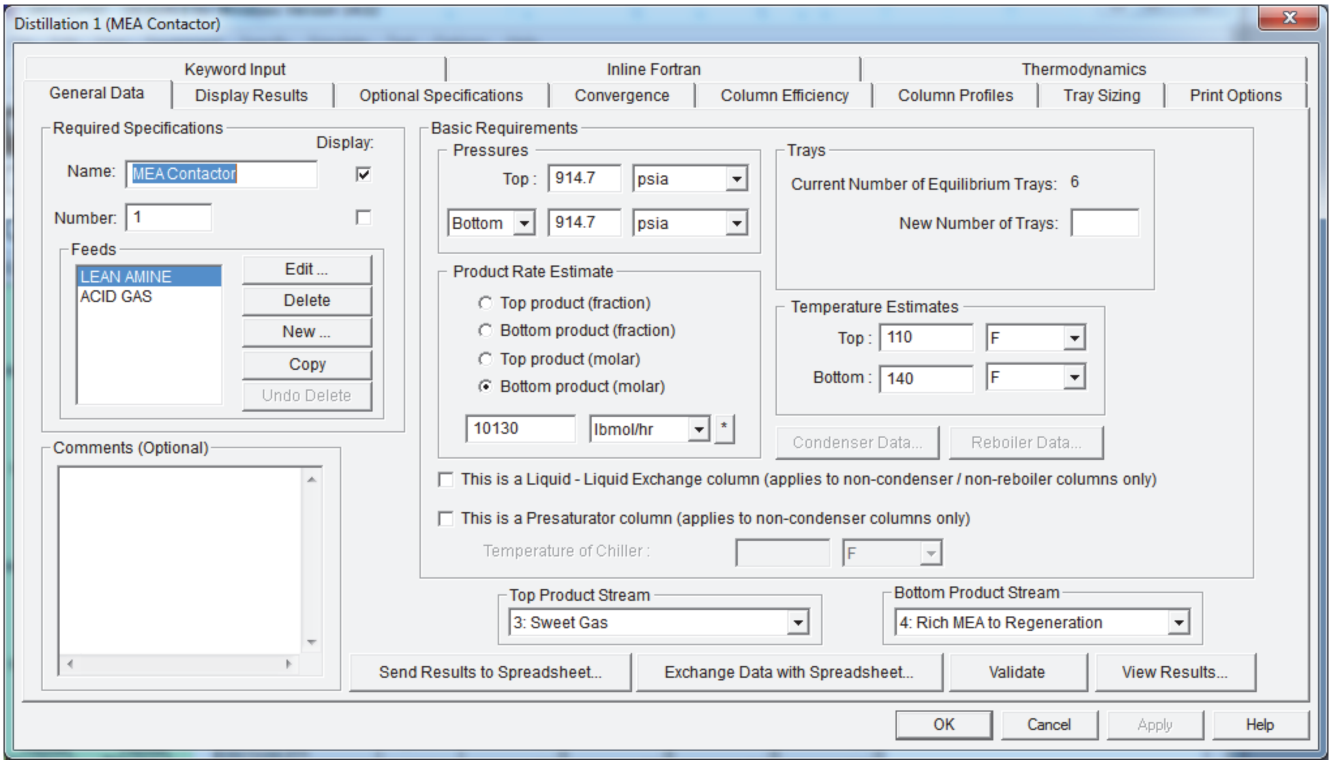


Figure 1: Distillation Dialog (from distl1.psd)

General Data

Data Item	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Feeds	Enter feeds to use. Click the New button to display the Feed - Specifications dialog that you can use to create new feed specifications. Select an existing feed and click the Edit button to modify the feed specifications. Select an existing feed and click the Delete button to remove that feed. Select an existing feed and click the Copy button to replicate the feed. You can then use the Edit function to make any minor changes to the copied feed.
Display Results on Flowsheet	You can optionally display the following results on the flowsheet: Trays Reflux Ratio Top Temperature Condenser Duty Bottom Temperature Reboiler Duty If you select to display one or more of these results, enter the number of digits to display to the right of the decimal.

Pressures

This group box is used to enter pressure specifications for the column. A linear pressure profile will be generated for the trays in the column from these specifications. If you have a nonlinear profile; then use the Profiles dialog under Optional Specs.

- Top Enter the top tray pressure. If you have a nonlinear pressure profile, the complete Pressure Profile, pressure of every tray, can be entered (see the Profiles dialog).
- Bottom Enter the bottom tray pressure for the column.

Trays

You use this group box to enter the number of theoretical trays in the column (not including the condenser and reboiler, if present). The top tray is always tray 1. The trays are numbered from top-to-bottom. Do not count the condenser or reboiler in the number of trays. DESIGN II numbers the condenser as tray zero and the reboiler as one greater than the number of trays. A minimum of two theoretical trays is allowed.

- Current Number of Equilibrium Trays — This item reports to you the number of equilibrium trays you have specified for the column. If you have not entered the number of trays, the space will be blank.
- New Number of Trays — Enter the number of equilibrium trays for the column. If you change the number of trays in the column, you may need to make adjustments in other specifications, such as the location of sidedraws. When you change the number of trays a third edit box appears. This allows you to add trays in a particular section of the column.

Product Rate Estimates

This group box is used to enter estimates for column product rates. It consists of a set of radio buttons that are summarized under data items in the table below. You first select the button that is appropriate for your estimate ; then enter the estimate in the edit box provided. The units field will reflect your choice of buttons. You can only make one selection. The top or bottom product rate you do not enter will be calculated to obtain an overall material balance.

- Top Product (fraction) — Enter your estimate for the top product as a fraction of the total feed to the column.
- Bottom Product (fraction) — Enter your estimate for the bottom product as a fraction of the total feed to the column.
- Top Product (molar) — Enter your estimate for the molar flow rate of the top product.
- Bottom Product (molar) — Enter your estimate for the molar flow rate of the bottom product.

Temperature Estimates

This group box is used to enter estimates for the temperature at the top and bottom of the column. A linear initial column profile is estimated from these temperature guesses for most columns (except a quadratic profile is used for the reboiled absorbers). In some cases, where the temperature profile is far from linear, it may be better to estimate the top or bottom temperature different from the actual temperature. Specifically, addition (or removal) of heat to tray(s) with an intercooler or side heater/cooler can cause a nonlinear temperature profile. In cases where nonlinear column temperature profiles are expected, a temperature profile for every tray in the column can be entered and may be required for convergence (see the optional Temperature dialog under Optional/Profiles). To estimate the product temperatures consider running a Shortcut column or a Component Splitter. Both modules provide guesses for the product rates as well as the top and bottom temperatures.

- Top — Enter the top product temperature as an initial guess. If the Temperature of Condenser specification is also entered; then this entry is the top tray temperature.
- Bottom — Enter the bottom product temperature as an initial guess.

Condenser — For Distill 3 and 4, click this button to display the Condenser dialog.

Reboiler — For Distill 2 and 4, click this button to display the Reboiler dialog.

This is a Liquid - Liquid Exchange Column — Select this checkbox to indicate that the distillation column is a liquid-liquid exchange column; this option only applies to non-condenser or non-reboiler columns.

This is a Presaturator Column (applies to non-condenser columns only) — Select this checkbox to indicate that the distillation column is a presaturator column.

Temperature of Chiller — If you select that this is a presaturator column, enter the temperature of the chiller and select a unit. The top feed location is specified as tray zero. The regular convergence technique will be used.

Top Product Stream — Select the stream to use for the top product.

Bottom Product Stream — Select the stream to use for the bottom product.

Feed

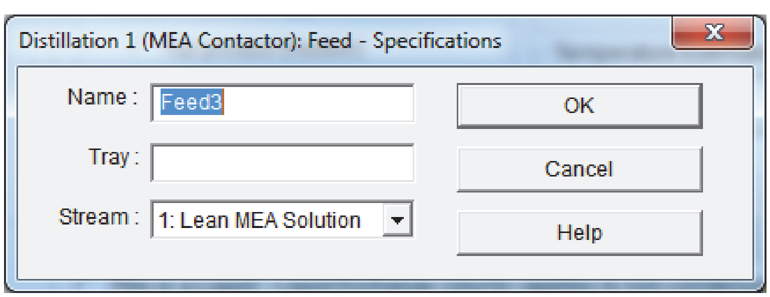


Figure 2: Feed Dialog (from distl1.psd)

Feeds can enter on any theoretical tray in the column. The top tray is always tray one and the condenser, if present, is tray zero. Any liquid in the feed will go on the feed tray. This means if the feed is two phase at the tray pressure; then the liquid portion of the feed enters the feed tray and the vapor portion automatically mixes with the liquid in the tray above. If the feed is all vapor; then all of the feed enters and mixes with the liquid in the tray above the feed tray. Feeds are always adiabatically flashed at the tray pressure. If the feed pressure is less than the tray pressure, calculations will continue but a warning message will state that a feed pump is required.

Data Item	Description
Name	view the name of the feed or change it.
Tray	Enter the tray location for the feed. The top tray in the column is always tray number one.
Stream	Select the flowsheet stream, from the drop-down list, that represents the feed.

Condenser Data

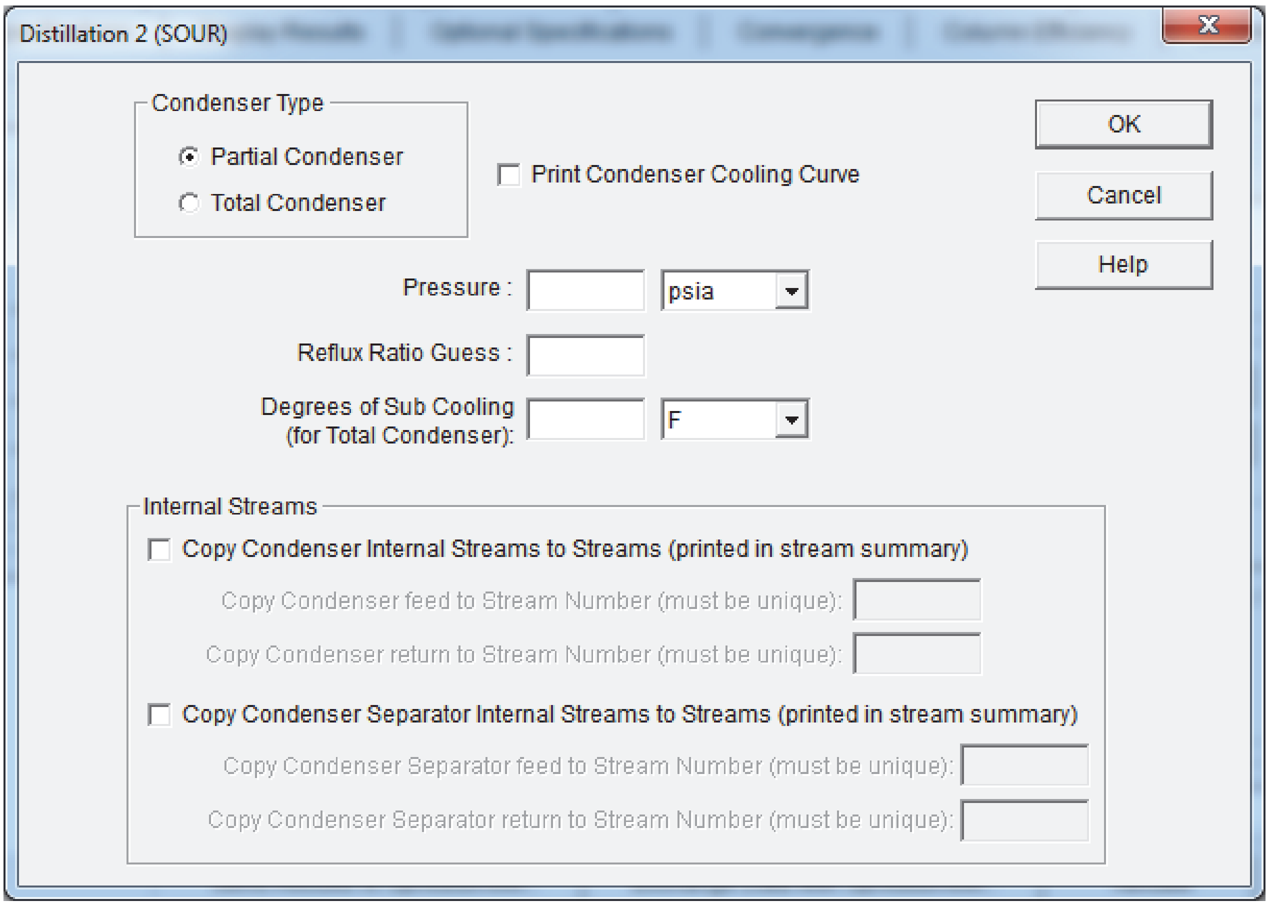


Figure 3: Condenser Data Dialog (from distl6.psd)

This dialog allows you to enter data describing the condenser for the column. If you wish to set condenser specifications, you will need to select the Main Specs dialog in the Required Specs menu for distillation.

Data Item	Description
Partial Condenser	Select this radio button if you want a partial condenser.
Total Condenser	Select this radio button if you want a total condenser.
Print Condenser Cooling Curve	Select this check box if you want a cooling curve produced for the vapor leaving the top tray of the column. The temperature range of the cooling curve is from the top tray temperature to the condenser outlet temperature. If you enter the pressure out of the condenser, equal pressure increments will be used for the cooling curve.
Pressure	Enter the pressure out of the condenser in this field.
Reflux Ratio Guess	Enter an initial guess for the ratio of moles of liquid returned on tray one to the total overhead product plus condenser decant, if any.

Degrees of Subcooling	Enter the number of degrees of subcooling for the condenser product liquid. This is only applicable for total condensers.
Internal Streams	
Copy Condenser Internal Streams to Streams	Select this checkbox to add internal stream data for the condenser to a stream, so that the data will appear in the stream summary output. Copy Condenser feed to Stream Number (must be unique): Enter a stream number. Copy Condenser return to Stream Number (must be unique): Enter a stream number.
Copy Condenser Separator Internal Streams to Streams	Select this checkbox to add condenser separator internal stream data for the condenser to a stream, so that the data will appear in the stream summary output. Copy Condenser Separator feed to Stream Number (must be unique): Enter a stream number. Copy Condenser Separator return to Stream Number (must be unique): Enter a stream number.

Reboiler Data

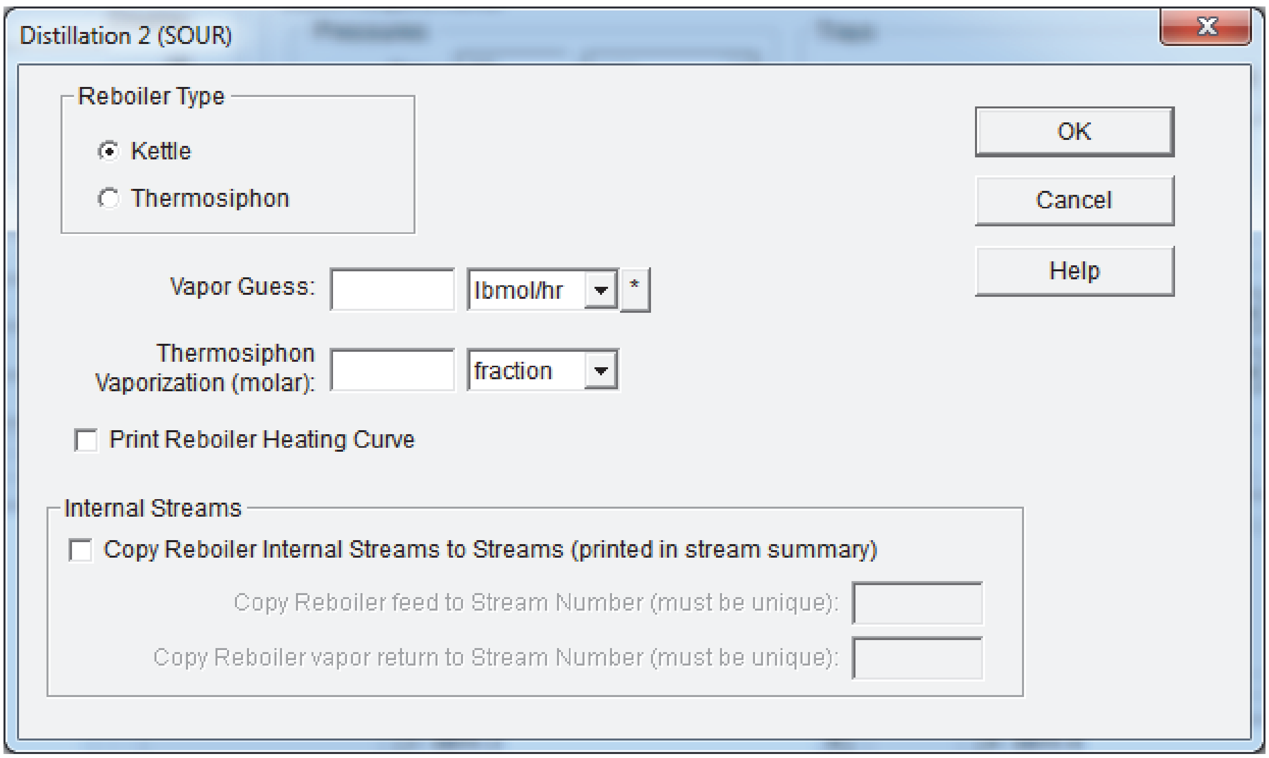


Figure 4: Reboiler Dialog (from distl6.psd)

This dialog allows you to enter data describing the reboiler for the column. You may select one of two types of reboilers: the kettle or thermosiphon type. If you wish to set reboiler specifications, you will need to select the Main Specs dialog in the Required Specs menu for Distillation.

Thermosiphon reboiler calculations are based on the following three assumptions:

- No mixing of return vapor and downcomer liquid
- Pump "work" input is zero
- No heat losses in the reboiler system

The bottom product will have the same composition as the circulating fluid. The product temperature will be between the bottom tray temperature and the returning fluid temperature. You can determine the circulation rate with the equation:

F = V / THE where
F = total flow to reboiler after bottom product is removed,
V = vapor returned to column in molar units,
THE = mole fraction vaporized.

The composition, but not the rate of the circulating fluid, is reported in the last stage of the tray-by-tray composition summary. PRInt SWItch = 2 is required to obtain this output.

Data Item	Description
Kettle	Select this radio button if you want a kettle reboiler.
Thermosiphon	Select this radio button if you want a thermosiphon reboiler.

Vapor Guess	Enter a guess for the vapor rate leaving the reboiler.
Thermosiphon Vaporization (molar)	Enter the mole fraction of the thermosiphon reboiler outlet stream that is vaporized.
Print Reboiler Heating Curve	Select this check box if you want a heating curve produced for the material in the reboiler. The temperature range of the heating curve is from the bottom tray temperature to the reboiler outlet Temperature

.

Internal Streams

Copy Reboiler Internal Streams to Streams	Select this checkbox to add internal stream data for the reboiler to a stream, so that the data will appear in the stream summary output. Copy Reboiler feed to Stream Number (must be unique): Enter a stream number. Copy Reboiler return to Stream Number (must be unique): Enter a stream number.

Main Specifications

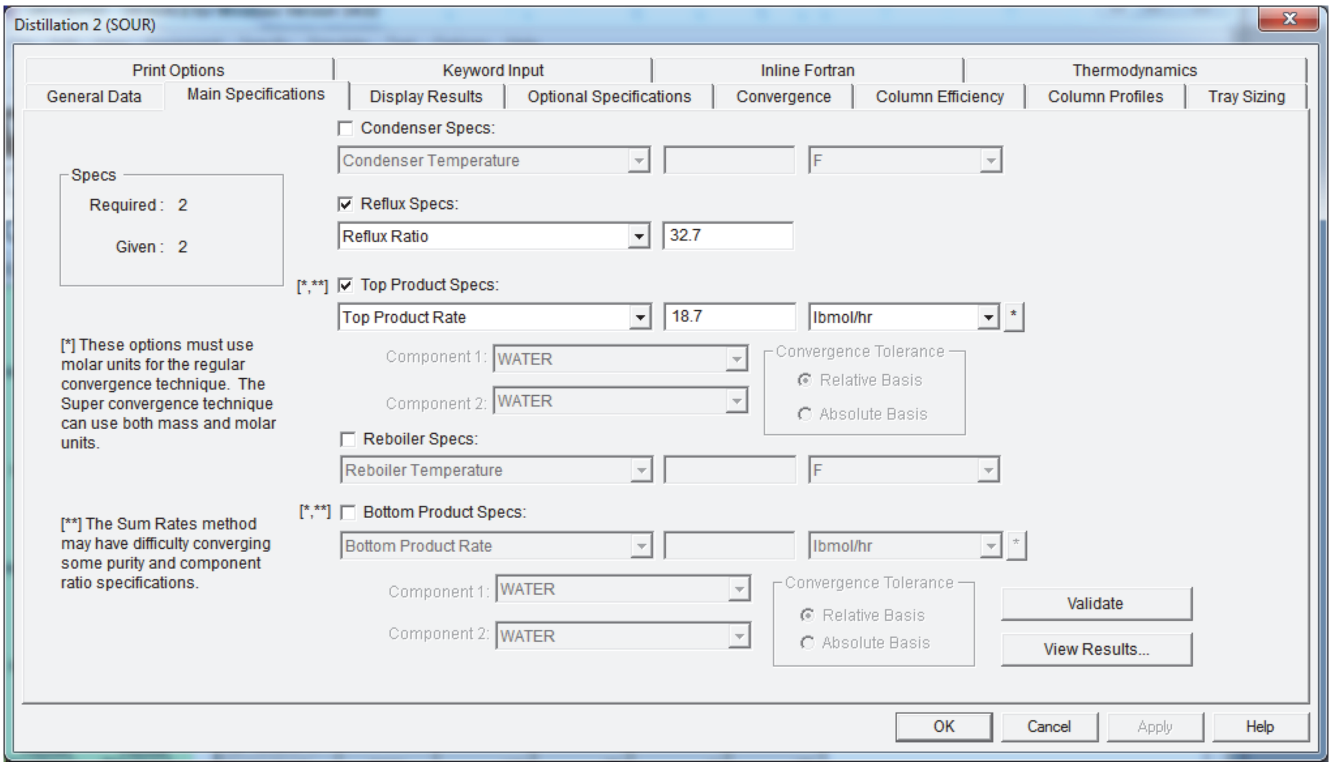


Figure 5: Main Specifications (from distl6.psd)

This tab is organized into a group box, check boxes, combo boxes, and cascading dialog buttons that you can use to make specifications on column parameters. The number of parameter specifications you can make depends on your column configuration. The group box labeled Specs tells you how many specifications you are allowed to make and displays a count of the ones you have made.

To set a column parameter, select a check box. You may obtain more detail about each of the check boxes by selecting from the items below. This dialog will allow you to select as many check boxes as you have parameter specifications to make for the column. When you attempt to select more than this number, one of the check boxes will be turned off. However, any specifications you have made under that check box will be retained and reactivated if you select that check box again.

Condenser Specs

The data items listed in the Condenser Specifications table appear when you select the Condenser Specs check box and select the corresponding combo box. When you select a data item for your condenser specification the appropriate dimensional units combo box will appear next to the edit box.

Data Item	Description
Condenser Temp	Enter the condenser temperature.
Condenser Duty	Enter the condenser duty. You should use the Super or Super Plus convergence option with this specification.

Reflux Ratio	Enter the ratio of the reflux to the total overhead product. The total overhead product is the sum of the vapor distillate and the liquid distillate plus decant, if any. If you do not intend to use the reflux as a specification for a column parameter, a guess can be entered for the reflux ratio on the Basic/Condenser dialog.
Reflux Flowrate	Enter the molar, mass, or volume rate of the liquid returning to tray one from the condenser. This option will require the Super (this is the default) or Super Plus convergence option. Super Plus does not handle this specification for stripper columns.

Top Product Specs

The data items listed in the Top Product Specs table appear when you select the Top Product Specs check box and select the corresponding combo box. When you select a data item for your top product specification the appropriate dimensional units combo box will appear next to the edit box.

Some of the top product specifications require the identification of chemical components.

Data Item	Description
Top Product Rate	Enter the total product rate for the top product. You should not specify both the top and bottom product rates.
Top Product Frac	Enter the total product for the top product as a fraction of the total feed to the column. You should not specify both the top and bottom product fractions.
Top Purity (mole)	Enter the fraction in molar units of a chemical component that you want in the overhead product. Enter the identity of the chemical component.
Top Purity (mass)	Enter the fraction in mass units of a chemical component that you want in the overhead product. Enter the identity of the chemical component.
Top Comp Rate	Enter the flowrate of a chemical component that you want in the overhead product. Enter the identity of the chemical component.
Top Comp Ratio (mole)	Enter the ratio of the flowrates in molar units for two chemical components that you want in the overhead product. Enter the identities of the two chemical components.
Top Comp Ratio (mass)	Enter the ratio of the flowrates in mass units for two chemical components that you want in the overhead product. Enter the identities of the two chemical components.

Reboiler Specs

The data items listed in the Reboiler Specifications table appear when you select the Reboiler Specs check box and select the corresponding combo box. When you select a data item for your reboiler specification the appropriate dimensional units combo box will appear next to the edit box.

Data Item	Description
Reboiler Temp	Enter the reboiler outlet temperature. Except for reboiled absorbers, you should use the Super or Super Plus convergence option with this specification.
Reboiler Duty	Enter the reboiler duty.

Bottom Product Specs

The data items listed in the Bottom Product Specs table appear when you select the Bottom Product Specs check box and select the corresponding combo box. When you select a data item for your bottom product specification the appropriate dimensional units combo box will appear next to the edit box.

Some of the bottom product specifications require the identification of chemical components.

Data Item	Description
Bottom Product Rate	Enter the total product rate for the bottom product. You should not specify both the top and bottom product rates.
Bottom Product Frac	Enter the total product rate for the bottom product as a fraction of the total feed to the column. You should not specify both the top and bottom product fractions.
Bottom Purity (mole)	Enter the fraction in molar units of a chemical component that you want in the bottom product. Enter the identity of the chemical component.
Bottom Purity (mass)	Enter the fraction in mass units of a chemical component that you want in the bottom product. Enter the identity of the chemical component.
Bottom Purity (volumetric)	Enter the fraction in volume units of a chemical component that you want in the bottom product. Enter the identity of the chemical component.
Bottom Comp Rate	Enter the flowrate of a chemical component that you want in the bottom product. Enter the identity of the chemical component.
Bottom Comp Ratio (mole)	Enter the ratio of the flowrates in molar units for two chemical components that you want in the bottom product. Enter the identities of the two chemical components.

Bottom Comp Ratio (mass)	Enter the ratio of the flowrates in mass units for two chemical components that you want in the bottom product. Enter the identities of the two chemical components.
Bottom Comp (volumetric)	Enter the ratio of volume fractions for two chemical components. Enter the identities of the two chemical components.
Bottom Prod Reid Vapor	Enter the Reid Vapor Pressure ASTM value. Enter the identity of the chemical component.

Optional Specifications

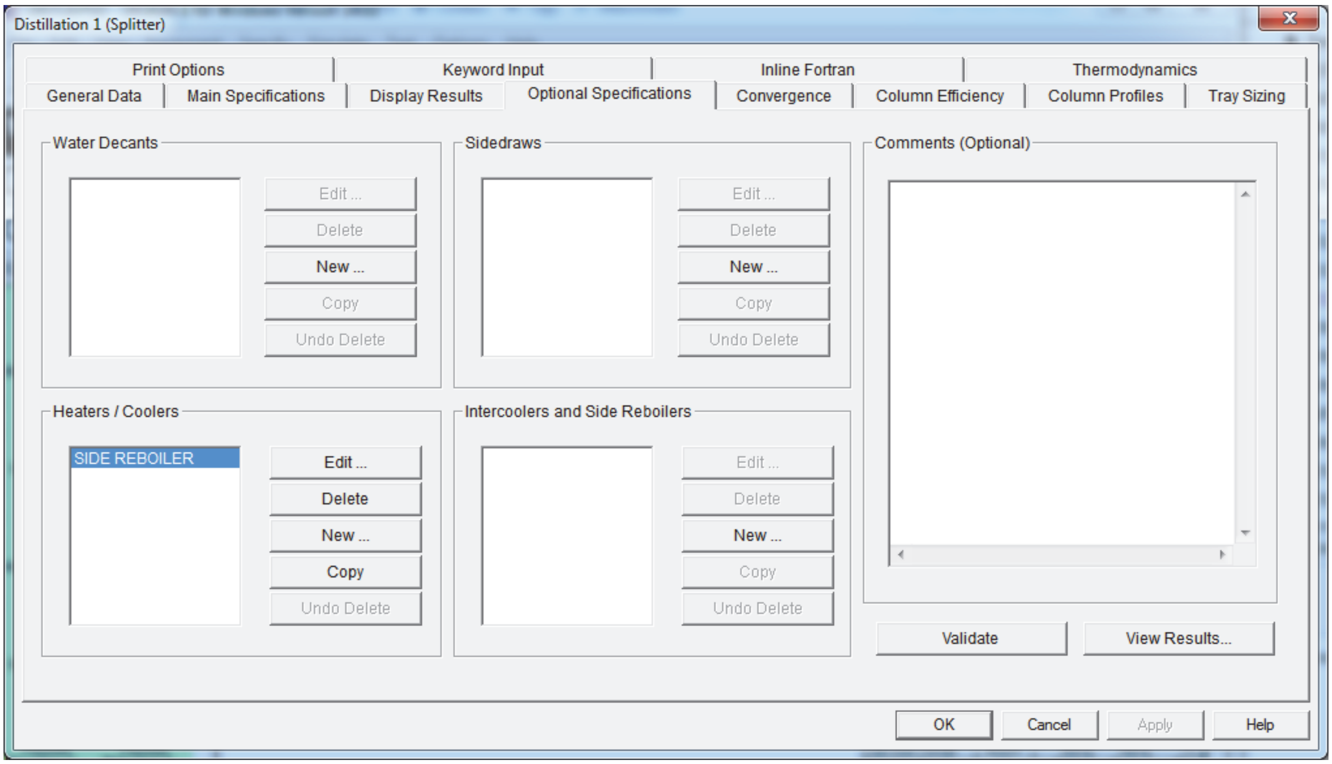


Figure 6: Optional Specifications Tab (from distl9.psd)

Data Item	Description
Water Decants	You can add, edit, delete or copy Water Decants.
Sidedraws	You can add, edit, delete or copy Sidedraws.
Heaters/Coolers	You can add, edit, delete or copy Heaters/Coolers.
Intercoolers	You can add, edit, delete or copy Intercoolers.

Optional Specifications: Water Decant

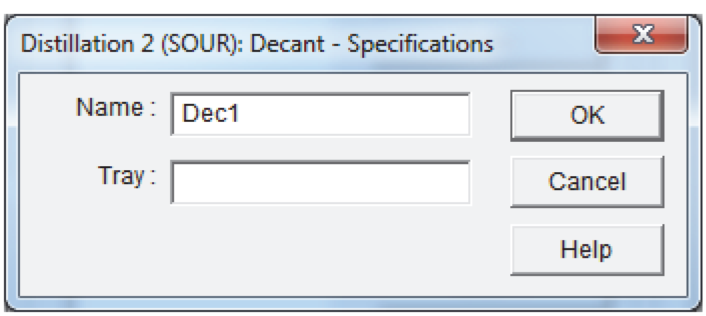


Figure 7: Decant – Specifications (from distl6.psd)

You can decant water from a tray or the condenser of the distillation column, to indicate what tray on which the decant occurs, and specify the amount of the decant

Water can be decanted from the condenser and/or any tray in the column. The decanted streams are NOT numbered on the flowsheet. To decant water from the condenser, use a tray number of zero. The Super or Super Plus equation solving method is required for decant specifications. A rigorous material and heat balance calculation is done for water on all trays of the column. Water solubility in the second liquid phase will be calculated by the methods specified with the Advanced Thermo dialog (accessed from the Basic Thermo dialog under the Specify menu).

Decant Tray Specifications

Data Item	Description
Name	Enter the name of the decant. If no entry is made, the default name that appears here will be used for the decant name.
Tray	Enter the tray location for the decant. The top tray in the column is always tray number zero.

Optional Specifications: Sidedraws

You can assign flowsheet streams as sidedraws to the distillation column, to assign the sidedraw to a column tray, and set specifications for the sidedraw.

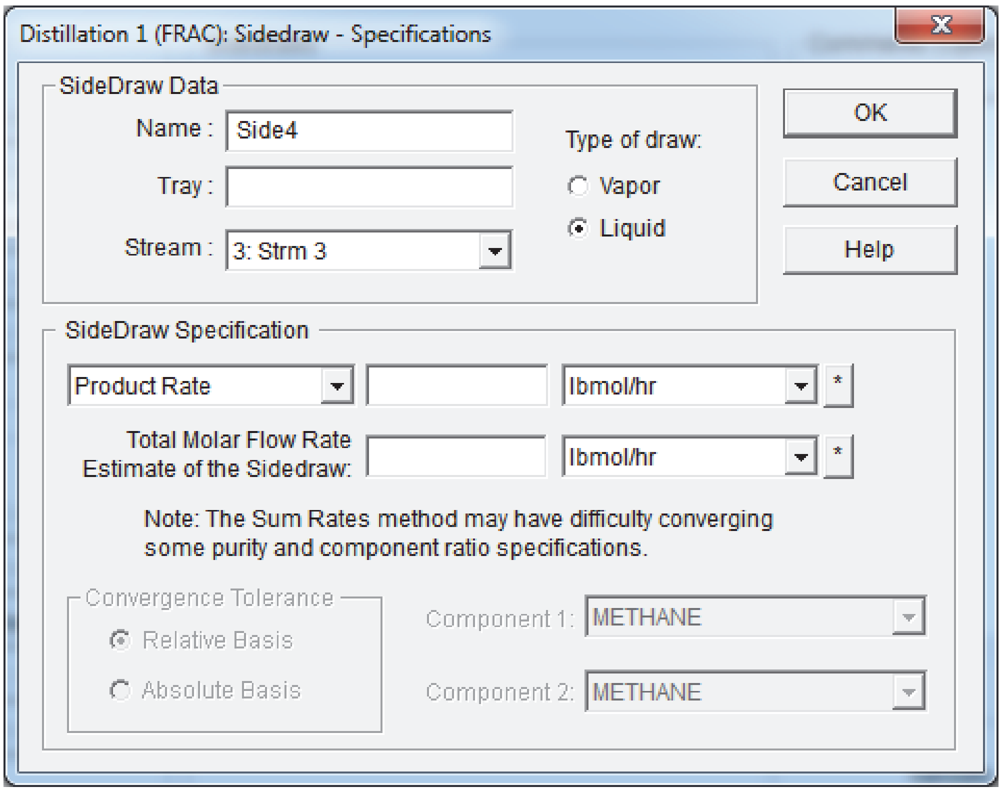


Figure 8: Sidedraw – Specifications (from distl7.psd)

Sidedraw rates are normally fixed. They will be varied only to meet a specification for the recovery of a particular chemical or a total flowrate for the sidedraw. Outside the distillation module the sidedraw rate can be varied with a Controller module or with Inline Fortran.

Data Item	Description
Name	Enter the name for the sidedraw. If the default name in the edit box is acceptable, no action is required.
Tray	Enter the tray number for the location of sidedraw.
Stream	Select the stream to use for the sidedraw from the drop-down list.
Vapor	Select this radio button for a vapor sidedraw.
Liquid	Select this radio button for a liquid sidedraw.
Product Rate	Enter the total flow rate for the sidedraw. If you select a non-molar flowrate; then the Flow Rate Estimate edit box appears. The molar flow rate will be adjusted by DESIGN II to satisfy the non-molar flowrate specification.
Comp Rate	Enter the flow rate for the recovery of a specific chemical component in your flowsheet. Selection of this option will cause the Flow Rate Estimate edit box to appear. It will also cause the Specify dialog button to appear that you use to indicate the specific chemical for the recovery.

Total Molar Flow Rate Estimate	Enter an initial guess for the molar flow rate that DESIGN II will use to meet a non-molar flow rate specification or a product recovery specification.

Optional Specifications: Heaters/Coolers

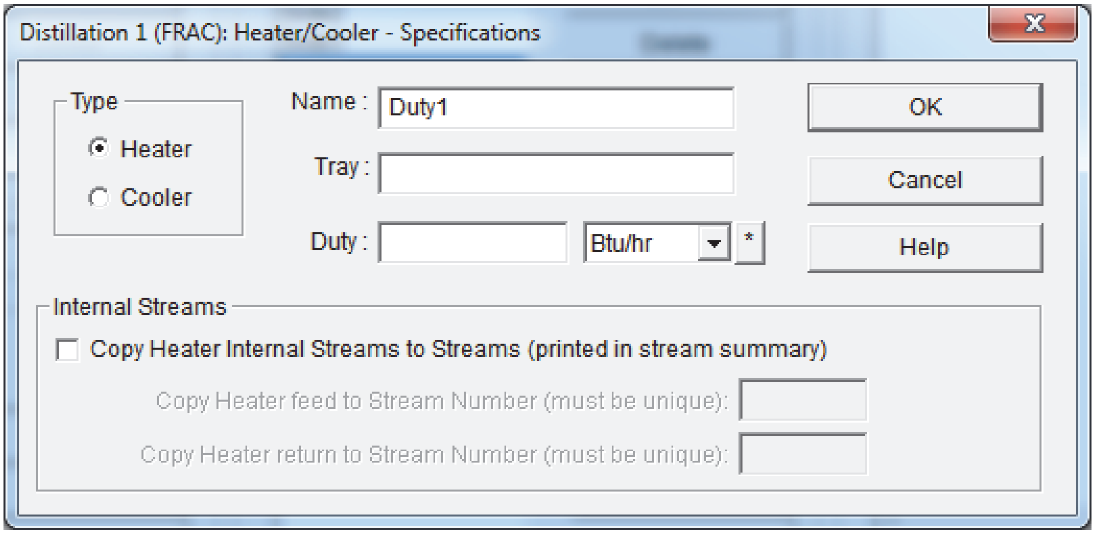


Figure 9: Heater/Cooler – Specifications (from distl7.psd)

You can create heaters and coolers for the Distillation column, to indicate what tray on which the heater/cooler occurs, and set the duty of the heater/cooler.

The duty that you enter is applied directly to the tray that you specify. Liquid and vapor on the tray are mixed and the duty is added or subtracted. Finally, the temperature corresponding to the new enthalpy is calculated along with the amount of vapor and liquid.

Data Item	Description
Heater	Select this radio button to set up a heater for a column tray.
Cooler	Select this radio button to set up a cooler for a column tray.
Name	Enter the name for the heater/cooler. If the default name in the edit box is acceptable, no action is required.
Tray	Enter the tray number for the location of the heater/cooler.
Duty	Enter the duty for the heater/cooler. Always enter a positive number since the radio button determines whether or not the duty will be used for heating or cooling.
Internal Streams	
Copy Heater Internal Streams to Streams	Select this checkbox to add internal stream data for the heater to a stream, so that the data will appear in the stream summary output. Copy Heater feed to Stream Number (must be unique): Enter a stream number. Copy Heater return to Stream Number (must be unique): Enter a stream number.

Optional Specifications: Intercoolers and Side Reboilers

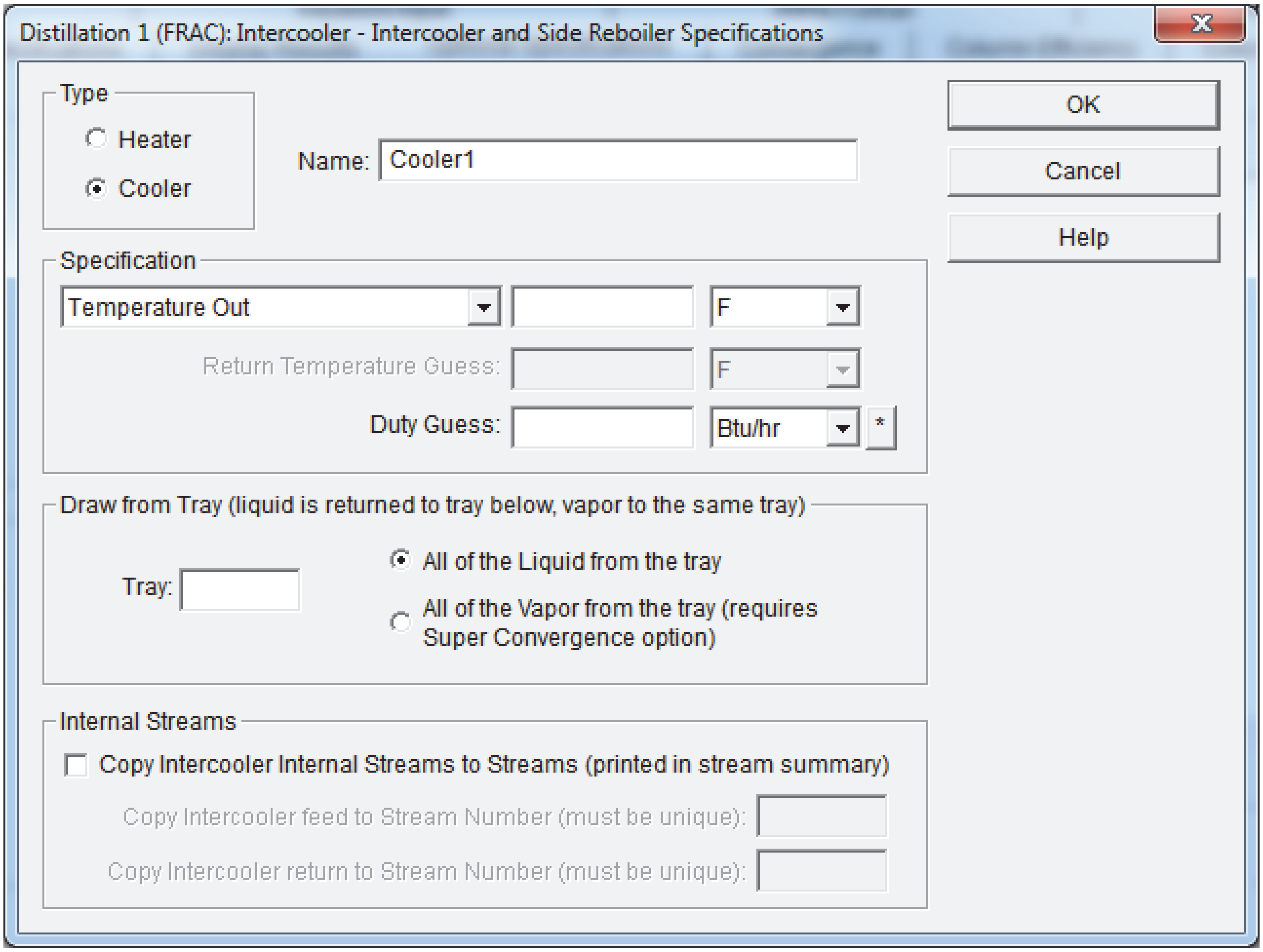


Figure 10: Intercooler – Intercooler and Side ReboilerSpecifications (from distl7.psd)

Use intercoolers/side reboilers to selectively cool or heat either ALL the liquid or ALL the vapor leaving a tray to a specified temperature and return it to the column. Enter only two commands to define all intercoolers. Always enter the location of intercoolers command and enter one of the cooling specifications: TEM INT, DUT INT, or DEL TEM INT.

Data Item	Description
Type	Select either heater or cooler for a column tray.
Name	View the name of the intercooler or change it.
Specification	
Temperature out	Enter the temperature(s) out of intercooler(s)/side reboiler(s). Enter one value for each intercooler location, separated by commas, in top-to-bottom order.
Duty	Enter the duty(s) of intercooler(s). Enter one value for each intercooler/side reboiler location, separated by commas, in top-to-bottom order. Duty is positive for heating and negative for cooling. Requires SUPer convergence option.
Delta Temperature	Enter the temperature changes for intercooler(s)/side reboiler(s). Enter one value for each intercooler/side reboiler location, separated by commas, in top-to-bottom order. Requires super convergence option. Intercoolers and heaters can significantly affect tray temperature and vapor traffic. Entering temperature and/or vapor profiles may help convergence.
Return Temperature Guess	Enter an estimate for the intercooler return temperatures.
Duty Guess	Enter a guess for the heat added to or removed from each intercooler in top to bottom order.
Draw from Tray (liquid is returned to tray below,	Enter the tray number to specify that either the liquid (positive tray number) or the vapor (negative tray number) from that tray is to be

vapor to the same tray)	cooled or heated. Select one of the following: All of the Liquid from the tray All of the Vapor from the tray (requires Super Convergence option).
Internal Streams Copy Intercooler Internal Streams to Streams	Select this checkbox to add internal stream data for the heater to a stream, so that the data will appear in the stream summary output. Copy Intercooler feed to Stream Number (must be unique): Enter a stream number. Copy Intercooler return to Stream Number (must be unique): Enter a stream number.

Convergence

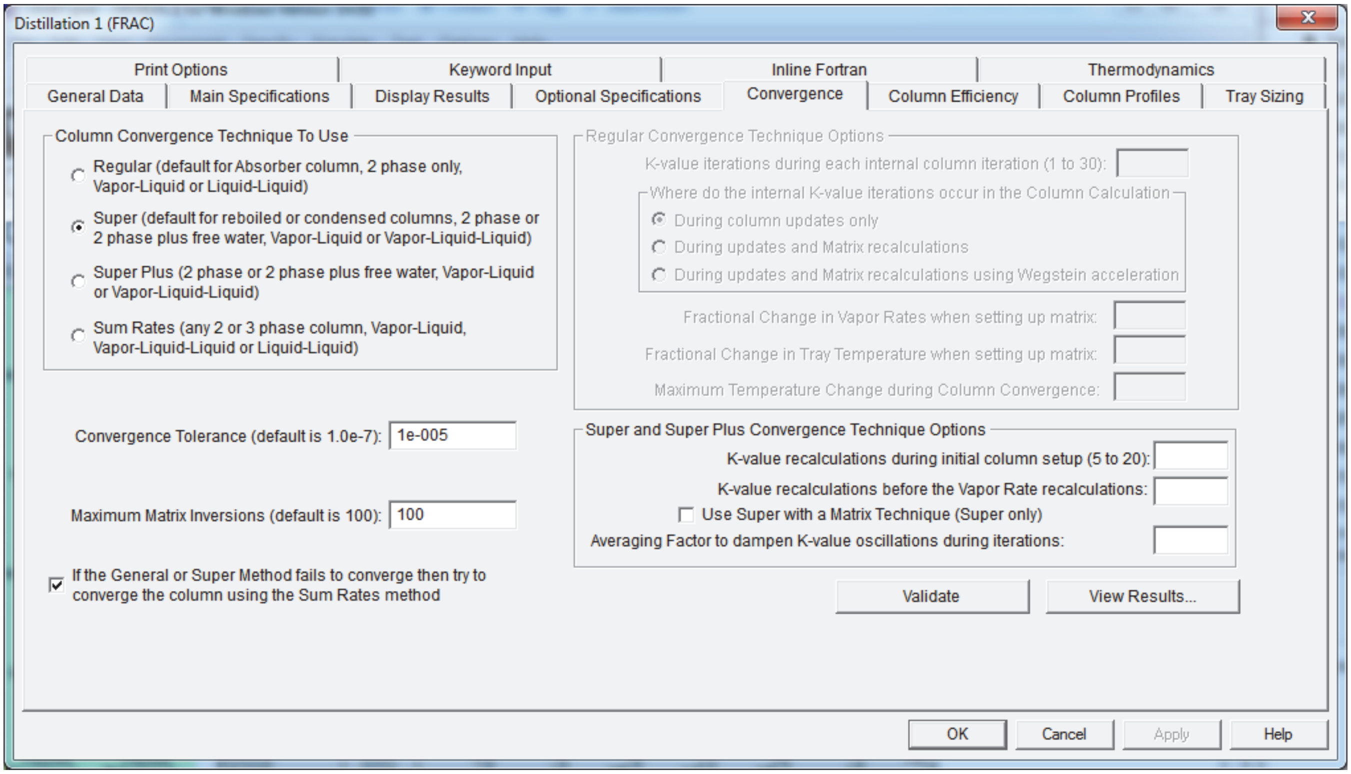


Figure 11: ConvergenceTab (from distl7.psd)

Data Item	Description
Column Convergence Technique to Use	Regular (default for absorber column): This is an equation solving technique that uses the temperature and vapor profiles as independent variables. It is the only option that can be used with absorbers. With this technique you have the following choices of specification variables: product rates, the condenser temperature, the reboiler duty and temperature, and the reflux ratio. This method will work better for wide boiling systems than Super. Super (default for reboiler or condensed columns): This is an equation solving technique that uses the temperature profiles as the primary external independent variables. Because of this formulation, it is in principle faster than Regular. With this technique you have the following choices of specification variables: product rates, condenser temperatures and duties, reboiler temperatures and duties, the reflux ratio, and product compositions. This technique generally works better for narrow boiling systems. Super PLUS: This is an extension of the Super technique. It improves its ability to solve distillation problems for wide boiling systems. Sum Rates (2 or 3 phase Absorbers (VL, VLL, or LLE)):
Regular Convergence Technique Options	K-value iterations during each internal column iteration (1 to 30): To stabilize compositional dependent K-values, enter the number of K-value iterations during each internal iteration. Where do the internal K-value iterations occur in the Column Calculation? • During column updates only • During udpates and Matrix recalculations • During updates and Matrix recalculations using Wegstein acceleration

	Fractional Change in Vapor Rates when setting up matrix: Enter the fractional change in vapor rates allowed when setting up column matrix. A range of 0.001 to 0.15 is recommended. For columns with high vapor to liquid ratios, this value can be calculated by dividing the minimum liquid rate by the average vapor rate; then dividing the answer by 2. Fractional Change in Tray Temperature when setting up matrix: Enter the fractional change in tray temperature allowed when setting up column matrix. A range of 0.001 to 0.15 is recommended. This value should be about half the Fractional Change in Vapor Rates. Maximum Temperature Change during Column Convergence: Enter the maximum temperature change to be used during column convergence. This number also represents the maximum percent change in the average column vapor rate allowed. For a more stable column convergence, normally for columns displaying high vapor rate dependence on temperature (cryogenic demethanizers), a value of 50 is recommended.
Convergence Tolerance	Enter the maximum acceptable NORM for the column. The NORM is the error in the tray heat and material balance on a fractional basis squared. The default is TOL = 0.1 E-4 or about 0.3% average error.
Maximum Matrix Inversions	Enter the maximum number of matrix inversions for column solution. Default is 100.
If the General or Super Method to converge then try to converge the column using the Sum Rates method	Select this to have the program attempt to converge the column using the Sum Rates fails method.
Super and Super PLUS Convergence Technique Options	K-value recalculations during initial column setup (5 to 20): To stabilize compositional dependent K-values, enter the number of times during initial column setup that K-values are recalculated in allowing for composition effects. This value should never exceed 20. K-value recalculations before the Vapor Rate recalculations: Enter how many of these K-value recalculations are performed before the vapor rate profiles are recalculated. This number should be about half the number entered for K-value recalculations during initial column setup. For columns with high compositional dependence of K-values, the number for K-value recalculations during initial column setup should be 10 and the number for K-value recalculations before the Vapor Rate recalculations should be 2. Use Super with a Matrix Technique (Super only): Select this checkbox to use the Super convergence with a matrix technique. This is normally not required. Averaging Factor to dampen K-value oscillations during iterations: Enter the averaging factor to dampen oscillations between iterations during convergence. For compositional dependent columns, a value of 0.5 is recommended and will cause averaging of the K-values from one iteration to the next. A value as small as 0.2 can be used. Small values of DVS slow convergence, but convergence may not be possible otherwise. For Super Plus: If this value is not entered; then the Super Plus technique automatically adjusts this parameter to enhance column convergence. The default value starts at 1 and is adjusted continuously during the column calculation. A value in the range of 0.2 to 0.999 can be entered to override the automatic adjustment.
Sum Rates (2 or 3 phase Absorbers only, VL, VLL, or LLE)	This is a three phase convergence method for Absorber Columns only. This also includes support for three phase liquid-liquid columns. The method does not support calculations when inter-coolers and tray heaters are present.

Optional Specifications: Column Efficiency

In order to approach more closely the actual operating conditions within a column, efficiencies for mass and heat transfer are needed for conventional column simulations. DESIGN II can perform a "modified vaporization efficiency" on the entire column. This is a simplified approach based on Holland & Welch's version ("Steam Batch Distillation Calculation," Petroleum Refiner, 36, No.5, 251 (1957)). They modified the original definition of "vaporization efficiency" by McAdams (Chemical Engineers Handbook, 3 ed., J. H. Perry).

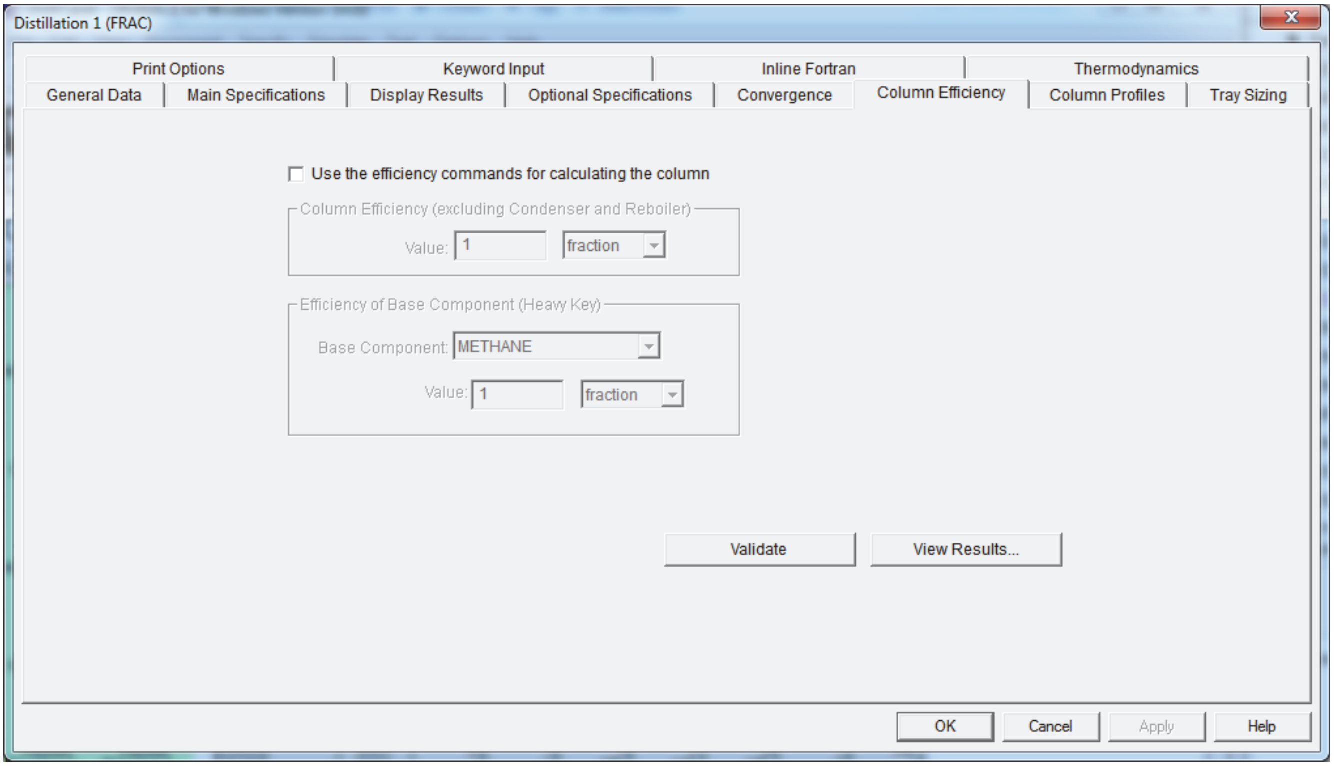


Figure 12: Column Efficiency Tab (from distl7.psd)

All commands are required and optional. If you do not want to use the efficiency commands, results will be based on the specified theoretical trays of the column.

Data Item	Description
Efficiency of the Column	Enter the desired column efficiency applied for all trays(except condenser & reboiler). Default is 1.0.
Efficiency of the base component i	Enter the efficiency of the base (heavy key) component in the mixture. "i" is the DESIGN II component ID number.

Notes:

1. The commands can be used only for DIStillation columns and not for REFine columns.
2. Both commands have to be used together (via Column Efficiency dialog or Keyword Input)
3. Only one "EFF BAS i" command and component can be used. Care should be exercised on the choice of heavy key component.

Optional Specifications: Column Profiles

In order to approach more closely the actual operating conditions within a column, efficiencies for mass and heat transfer are needed for conventional column simulations. DESIGN II can perform a "modified vaporization efficiency" on the entire column. This is a simplified approach based on Holland & Welch's version ("Steam Batch Distillation Calculation," Petroleum Refiner, 36, No.5, 251 (1957)). They modified the original definition of "vaporization efficiency" by McAdams (Chemical Engineers Handbook, 3 ed., J. H. Perry).

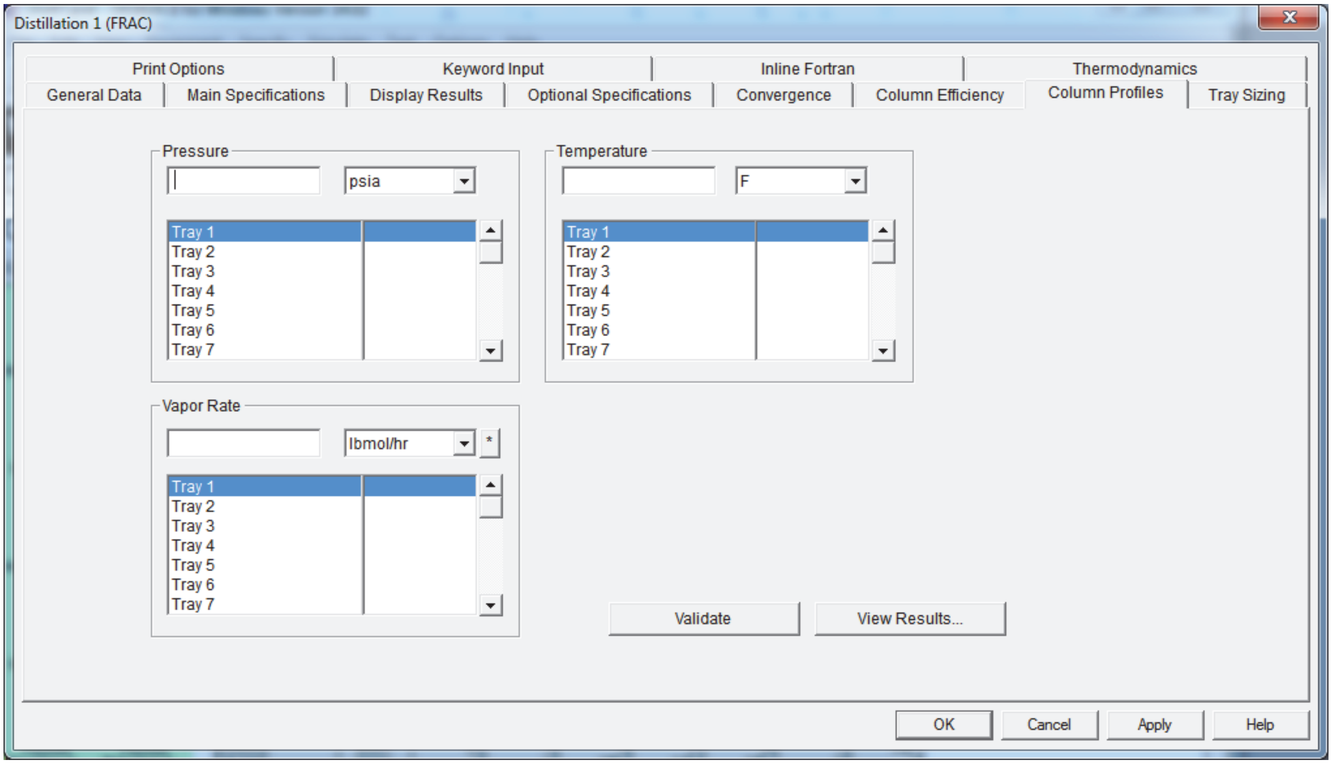


Figure 13: Column Profiles Tab (from distl7.psd)

Pressure

Using this section you can enter pressure profiles for the column. It is not necessary to enter the pressure for each tray in the column. If you enter a zero (or make no entry at all) for a tray pressure or a succession of tray pressures; then DESIGN II will perform a linear interpolation for the trays with zero pressures. If a nonlinear pressure profile is suspected or known to exist, the complete pressure of every tray can be entered. If you have not specified the number of trays for the column, nothing will appear in the left-most list box. If you have specified the number of trays; then a list of trays with their numbers will appear.

Temperature

Using this section you can enter temperature profiles for the column. It is not necessary to enter the temperature for each tray in the column. If you enter a zero (or make no entry at all) for a tray temperature or a succession of tray temperatures; then DESIGN II will perform a linear interpolation for the trays with zero temperatures. If a nonlinear temperature profile is suspected or known to exist, the complete temperature of every tray can be entered. If you have not specified the number of trays for the column, nothing will appear in the left-most list box. If you have specified the number of trays; then a list of trays with their numbers will appear.

Vapor Rate

Using this section you can enter vapor rate profiles for the column. It is not necessary to enter the vapor rate for each tray in the column. If you enter a zero (or make no entry at all) for a tray vapor rate or a succession of tray vapor rates; then DESIGN II will perform a linear interpolation for the trays with zero vapor rates. If a nonlinear vapor rate profile is suspected or known to exist, the complete vapor rate of every tray can be entered. If you have not specified the number of trays for the column, nothing will appear in the left-most list box. If you have specified the number of trays; then a list of trays with their numbers will appear.

Optional Specifications: Tray Sizing

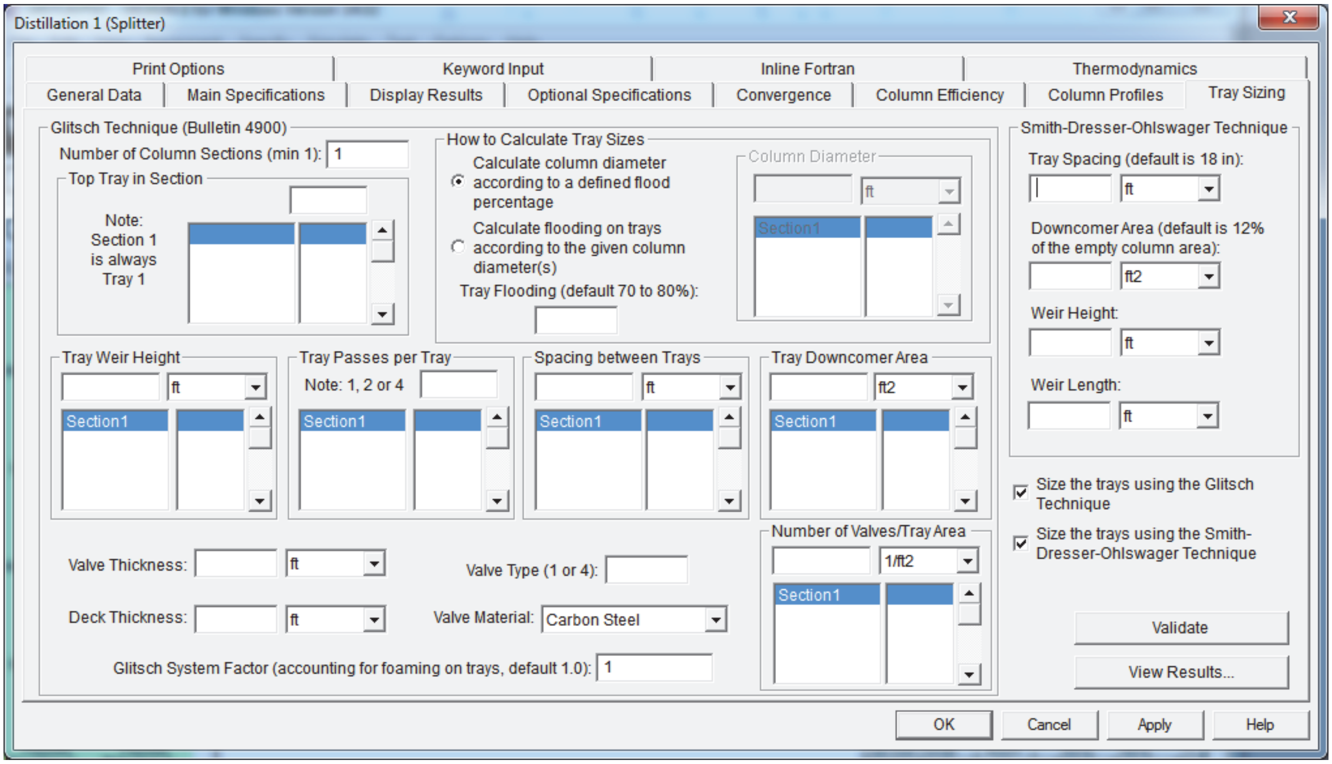


Figure 14: Tray Sizing Tab (from distl9.psd)

Optional Specifications: Distillation Tray Sizing

There are two different shortcut sizing techniques in the Distillation column calculations. Both calculations are performed automatically using default values unless you enter one or more of the data items listed below.

Smith-Dresser-Ohlswager Technique

The first is the Smith-Dresser-Ohlswager technique (Smith, R.B., Dresser, T. and Ohlswager, T., Hydrocarbon Processing, Vol. 40, No. 5., pp. 183- 184 (1963)). This correlation calculates the column diameter at 100 percent of flood.

All commands are optional. If you do not want to use the default 18 inches settling height, you must enter all of the following commands:

Data Item	Description
Tray Spacing	Enter the tray spacing to be used along with Weir Height and Weir Length in calculating the settling height. Default is 18 inches settling height.
Downcomer Area	Enter the downcomer area in square feet. Default is 12 percent of the empty column area.
Weir Height	Enter the Weir Height to be used along with Tray Spacing and Weir Length in calculating the settling height.
Weir Length	Enter the Weir Length to be used along with Tray Spacing and Weir Height in calculating the settling height.

Values entered with the Tray Spacing, Weir Height, and Weir Length data items are used in calculating the settling height (tray spacing minus clear liquid depth). If any one of these three data items is not entered, the settling height is assumed to be 18 inches. Minimum settling height is 2 inches and maximum is 30 inches for the correlation. Downcomer Area is used in calculating the vapor velocity on each stage. If not entered, the Downcomer Area is set to 12 percent of the empty column area.

Glitsch Technique

The second sizing technique is provided in cooperation with Glitsch Incorporated (Glitsch Bulletin 4900, Design Procedure for Ballast Trays). This shortcut method was developed for ballast trays but is also applicable for sieve trays with 14% hole area. This technique is applicable for tray spacings of 12-48 inches and percent of flood values from 20-100%.

By default the Glitsch, Inc. correlation calculates column tray diameters at 80% of flood based on tray loadings, system factor and tray spacing. The number of passes will be calculated, if you do not specify a value. You can specify a different percent of flood using the Glitsch Percent flood command.

Data Item	Description
Number of Column Sections	Enter the number of column sections to use. The minimum number is 1.
How to Calculate Tray Sizes	Select either: Calculate column diameter according to a defined flood percentage Calculate flooding on trays according to the given column diameter(s) If you selected Calculate column diameter according to a defined flood percentage: Tray Flooding (default 70 to 80%): Enter the percent of flood for the column. If you selected Calculate flooding on trays according to the give column diameter(s): Click on the name of a section from the list and enter a column diameter then select a unit. Repeat this for each section.
Tray Flooding	Enter the percent of flood for the column. If not specified, the program will calculate. One input value allowed.
Tray Weir Height	Click on the name of a section from the list and enter a weir height then select a unit. Repeat this for each section.
Tray Passes per Tray	Click on the name of a section from the list and enter the number of tray passes. One input value is allowed: 1, 2, or 4 passes.
Spacing between Trays	Enter the spacing(s) of trays. Default is 2 feet. The technique is valid for tray spacings from 1 to 4 feet.
Tray Downcomer Area	Enter the downcomer area in square feet. Default is 12 percent of the empty column area.
Number of Valves/Tray Area	Click on the name of a section from the list and enter the number of valves per square foot of active area and select a unit. Repeat this for each section.
Valve thickness	Enter the valve thickness and select a unit.
Valve Type	Enter the valve type. Either 1 or 4 is available (per Glitsch Bulletin 4900). 1 serves as a general purpose standard size unit, used in all services. The legs are formed integrally with the valve for deck thicknesses up to 3/8". 4 signifies a venturi-shaped orifice opening in the tray floor which is designed to reduce substantially parasitic pressure drop at the entry and reversal areas. A standard ballast unit is used in this opening, normally. The maximum deck thickness permissible with this opening is 10 gauge.
Deck Thickness	Enter the deck thickness and select a unit.
Valve Material	Select one of the following: Aluminum Carbon Steel Copper Monel Nickel 200 Hastalloy Stainless Steel 304 Lead Titanium
Glitsch System factor	Enter the system factor for foaming on the trays. Default is 1.0 which is appropriate for non-foaming, regular systems. One input value allowed.
Size the trays using the Glitsch Technique/Size the trays using the Smith-Dresser-Ohlswager Technique	You can select to size the trays using one, both, or neither technique (so no column sizing is performed). Check or un-check the box next to each technique.

Optional Specifications: Print Options

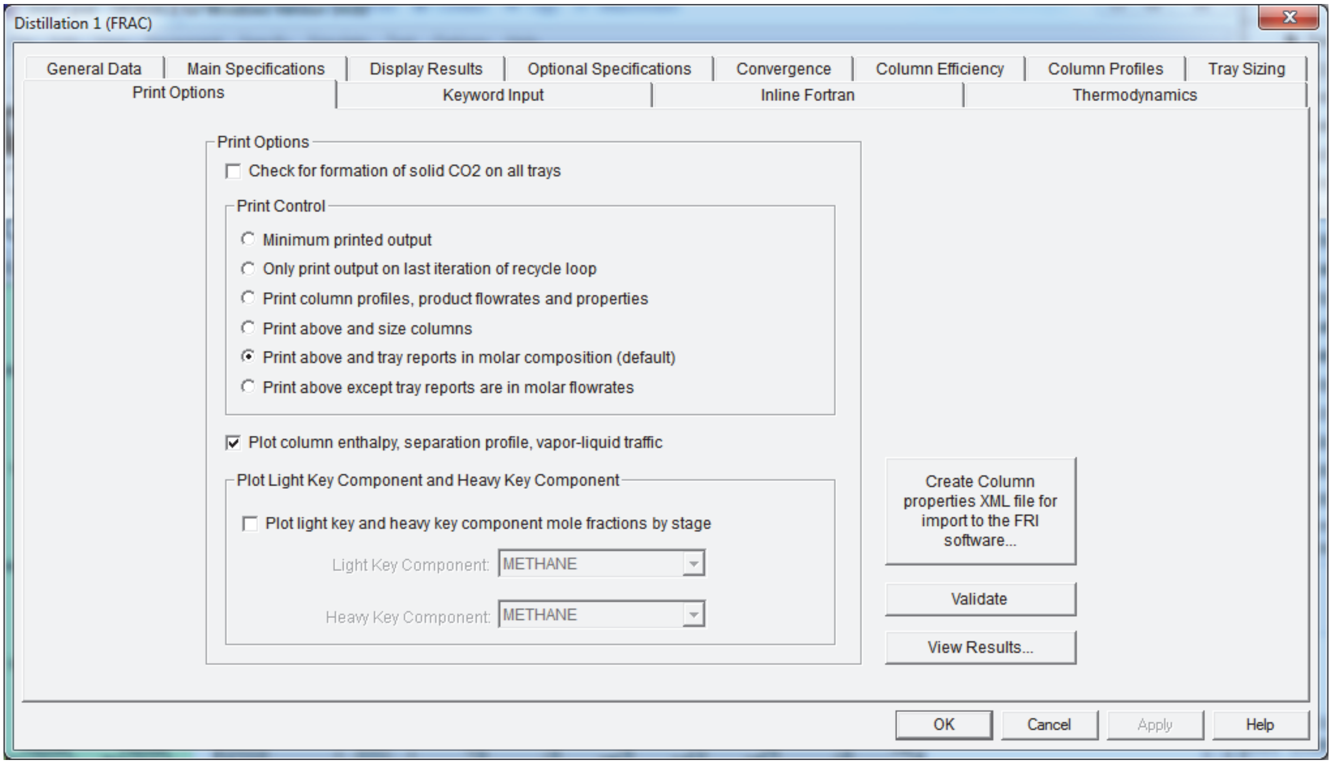


Figure 15: Print Options Tab (from distl7.psd)

Data Item	Description
Check for formation of solid CO2 on all trays	Use this to check whether solid CO2 will form at any of the tray conditions in the column (typically for cryogenic natural gas mixtures).
Print Control	Select one of the following: Minimum printed output Only print output on last iteration of recycle loop Print column profiles, product flowrates, and properties Print above and size columns Print above and tray reports in molar composition (default) Print above except tray reports are in molar flowrates
Plot column enthalpy, separation profile, vapor liquid traffic	Select this checkbox to print enthalpy versus temperature for reboiler and condenser curves, separation parameter profile, and vapor-liquid traffic profile.
Plot Light Key Component and Heavy Key Component	
Plot light key and heavy key component mole fractions by stage:	Click this checkbox to perform this option.
Light Key Component:	Select the light key component from the drop-down list to get a plot of light key component mole fractions in the vapor and liquid for each stage of the column.
Heavy Key Component:	Select the heavy key component from the drop down list.
Create Column properties XML file for import to the FRI software	Select this button to export column properties in a file format that can be imported into Fractionation Research Inc. software (see www.fri.org for more details about this software)

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II Reference Guide.

Distillation Examples

There are several sample flowsheets in “Chapter 17: Distillation Column Samples - c:\designii\samples\equipmnt\distill” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.11: Divider / Divider (Multiple)

General

The Divider divides a stream into two or more streams of the same composition, temperature, and pressure using multiple flowrate or fraction specifications. The divider (multiple) splits a stream into three or more streams of the same composition, temperature, and pressure.

Details

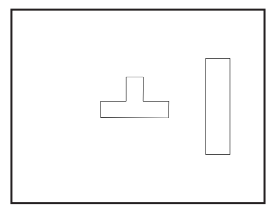
The Divider module is often used with the Controller to divert part of a stream to optimize heat exchange or component recoveries. If you enter only the fraction or flowrate for the first outlet stream, the Divider module automatically places the remainder in the second outlet stream. A Controller module can then be used to control the Divider by varying the fraction or flowrate of the first outlet stream. This technique works only when a Divider has two outlet streams. If the Divider you want to control has more than two outlets; then use the Multiple Divider.

Please see the online **DESIGN II** Help topic ***Equipment/Divider*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 12: Divider*** for more details.

Symbols

The Divider unit module has two symbols.

The Divider module requires that one or more inlet and two outlet streams be connected to the module. The Multiple Divider module requires that one or more inlet and two to six outlet streams be connected to the module.



Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

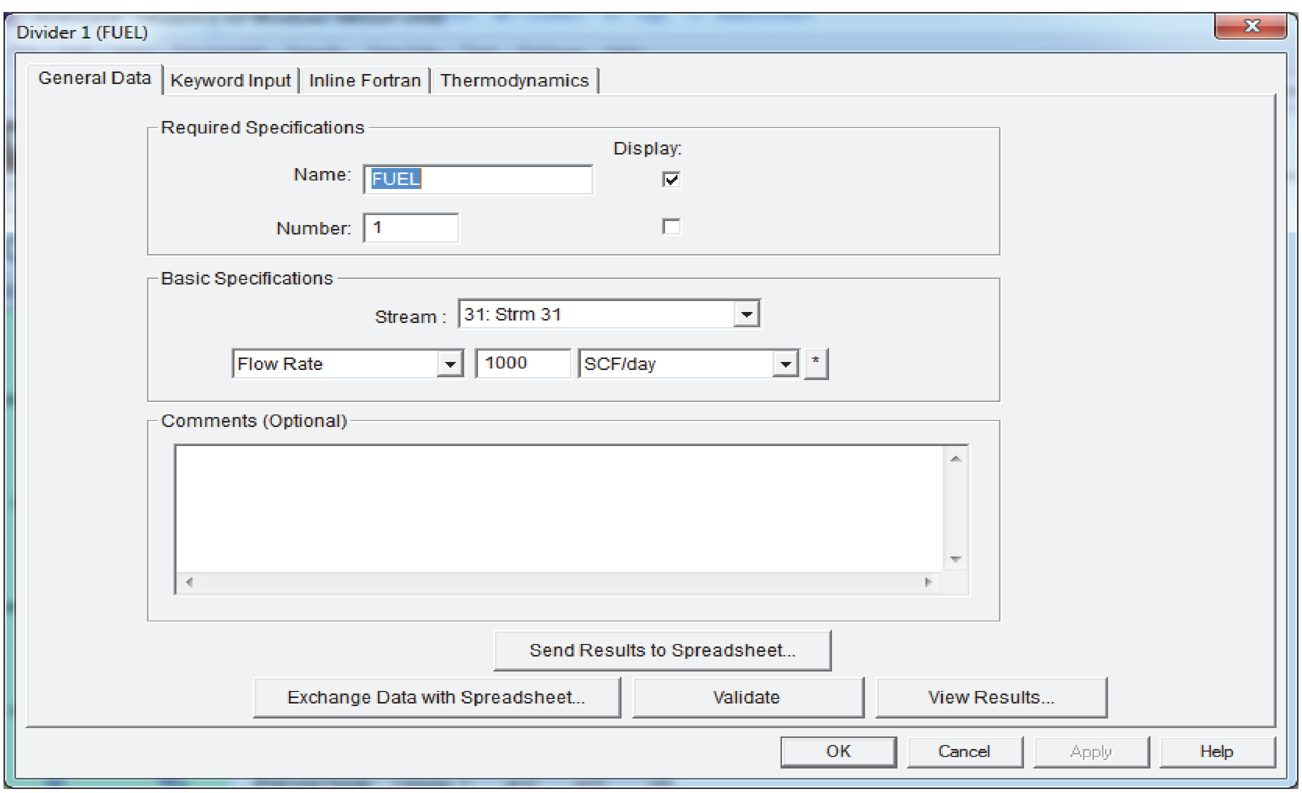


Figure1: Divider Dialog (from divdr2.psd)

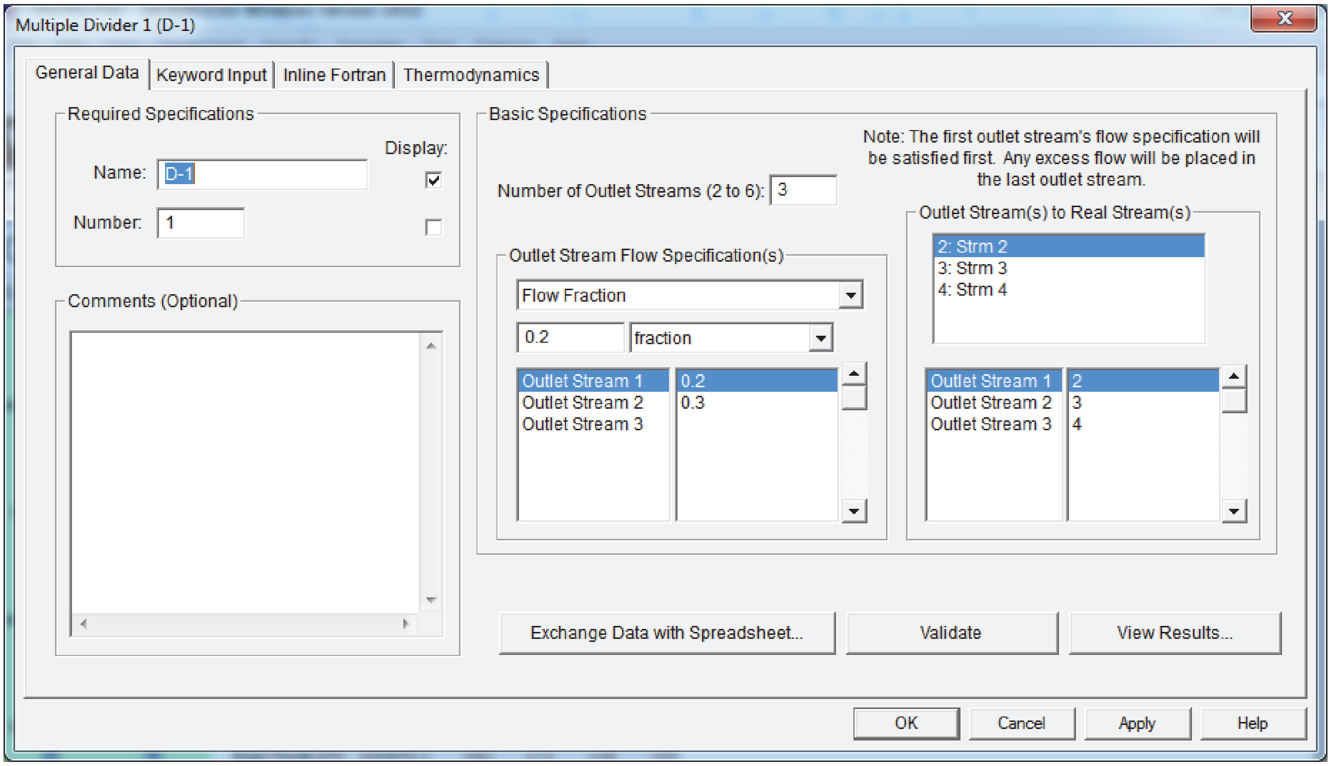


Figure 2: Multiple Divider Dialog (from divdr1.psd)

The multiple divider requires 1 to 8 inlet and 2 to 6 outlet streams be connected to the module.

General Data: Divider

Data Item	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Stream	Select the name of the stream you want to divide.
Flow Fraction/Flow Rate	Enter the flow fraction or flow rate of the inlet stream to be allocated to the output stream. If the feed is smaller than the flow rate specification, the divider will place all of the feed in the first outlet stream.

General Data: Divider (Multiple)

Data Item	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.

Note: The first outlet stream's flow specification will be satisfied first. Any excess flow will be placed in the last outlet stream.

Number of Outlet Streams	View or change the total number of outlet streams (between 2 and 6). The default is 2.
Outlet Stream(s) to Real Stream(s)	Click on a stream in the outlet stream list; then click on a stream from the list of real streams (directly below the outlet stream list). The outlet stream and real stream are matched.
Outlet Stream Flow Specifications	Select either: Flow Fraction Flow Rate Click on an outlet stream from the list. Then, enter the fraction/flow rate of the inlet stream to be allocated to the output stream and select a unit. Flow rate notes: 1) If the feed is smaller than the flow rate specification, the divider will place all of the feed in the first outlet stream. 2) To enter base quantity and time units, click the *.
Outlet Stream to Real Streams	Click on a stream in the outlet stream list; then click on a stream from the list of real streams (directly below the outlet stream list). The outlet stream and real stream are matched.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears on this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Divider Examples

There are several sample flowsheets in “Chapter 18: Divider Samples - c:\designii\samples\equipmnt\divider” of the DESIGN II for Windows Tutorial and Samples Guide.

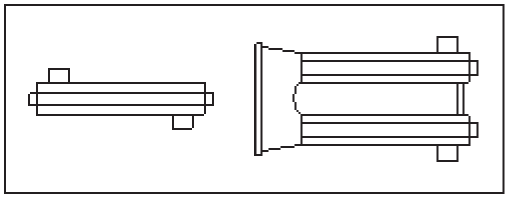
Chapter 6.12: Double Pipe Exchanger

General

The double pipe heat exchanger is a pipe within a pipe that exchanges heat between the inner pipe and outer pipe. This heat exchanger requires that two inlet and two outlet streams be connected to it.

Symbols

The Double Pipe Exchanger module has two symbols:



Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

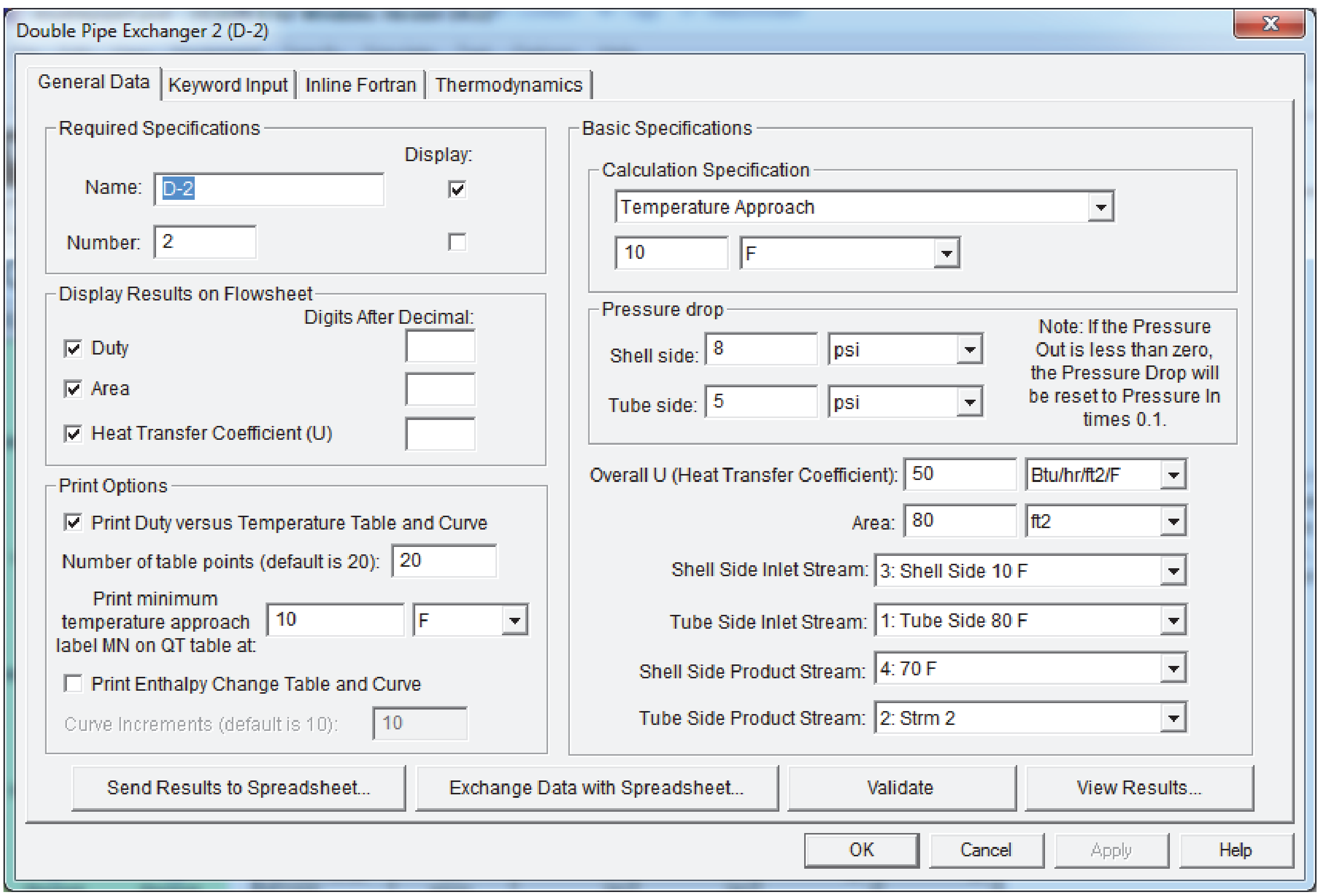


Figure 1: Double Pipe Exchanger Dialog (from doublepipe5.psd)

General Data

Data Item	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.

Display Results on Flowsheet	You can optionally display the following results on the flowsheet: Duty, Area, and Heat Transfer Coefficient (U). If you select to display one or more of these results, enter the number of digits to display to the right of the decimal.
Print Options	Select this option to print a Duty versus Temperature Table and Curve on the output. The Q-T table and plot are useful tools for evaluating exchanger design. These curves are based on end-point temperature results. You can quickly determine the minimum temperature approach or locate regions with unrealistically high U x A or temperature crossovers. In addition, bubble points and dew points are marked when vaporization or condensation occurs for either hot or cold streams.
Number of table points	Default is 20. Enter the number of points to use for plotting the curve.
Print minimum temperature approach label MN on QT table at:	Enter a temperature and select a unit. This will place the label "MN" on the Q-T table to indicate the minimum temperature approach, based on the value you entered.
Print Enthalpy Change Table and Curve	Select this to include an enthalpy change table and curve.
Calculation Specification	Select either: Temperature Out Shell Side: Enter the desired temperature of the outlet stream. Duty: Enter the desired enthalpy change of the input stream. A positive value indicates heating; negative indicates cooling. UA Exchanger: Enter the Overall U and Area in the fields below. Temperature Approach: Enter the absolute temperature difference between the shellside outlet stream and the tubeside inlet stream (must be greater than 2 °F). Shell Side Delta Temperature: Enter the desired change in temperature from shell side inlet to outlet. Positive value indicates heating; negative indicates cooling. Shell Side Temperature Out is Bubble Point: Sets the temperature of the shell side outlet stream corresponding to the shell side inlet stream's bubble point. Shell Side Temperature Out is Dew Point: Sets the temperature of the shell side outlet stream to the corresponding shell side inlet stream's dew point. Temperature Out Tube Side: Enter the temperature out of the tube side. After selecting a Calculation Option, enter the corresponding value and select a unit. To enter base quantity and time units, click the *.
Pressure drop Shell side/ Tube Side	Enter the shell side pressure drop and tube side pressure drop and select a unit. If the Pressure Out is less than zero, the Pressure Drop will be reset to Pressure In times 0.1
Overall U (Heat Transfer Coefficient	Enter the overall heat transfer coefficient and select a unit.
Area	Enter the heat transfer surface area per shell and select a unit.
Shell Side Inlet Stream/ Tube Side Inlet Stream/ Shell Side Product Stream/ Tube Side Product Stream	Open the drop down list and select the desired stream to use for the Shell/Tube Inlet/Product Stream.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears on this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Chapter 6.13: Equilibrium Reactor

General

This generalized equilibrium reactor module calculates gas-phase reactions of multi-component systems. Chemical equilibrium can be calculated isothermally or adiabatically. Isothermal calculations are the default. This reactor module is not restricted to a specific type of reaction. Heat and entropy of formation data at 25C are required for all components. This data is available automatically for the first 98 components in the Database Components.

Details

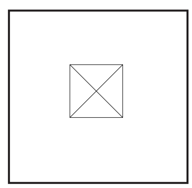
You can enter product constraints or stoichiometric coefficients for reactions occurring at conditions away from equilibrium such that extent of reaction or temperature of approach specifications can be used. You can use either the global reaction commands or the specific reaction commands to specify the products approach to equilibrium. If you do not enter any reaction commands, DESIGN II will automatically redistribute all atomic species among the specified molecular species to achieve the most thermodynamically stable molecular compositions. Since these are gas-phase reactions, one of the following equations of state techniques should be used:

APISOAVEK   LKPK   BWRK   PENK   BWRSK   RKK

Please see the online **DESIGN II** Help topic ***Equipment/Equilibrium Reactor*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 13: Equilibrium Reactor*** for more details.

Symbols

The Equilibrium Reactor module has one symbol:



The Equilibrium Reactor module requires that one or more inlet streams and one outlet stream be connected to the module.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

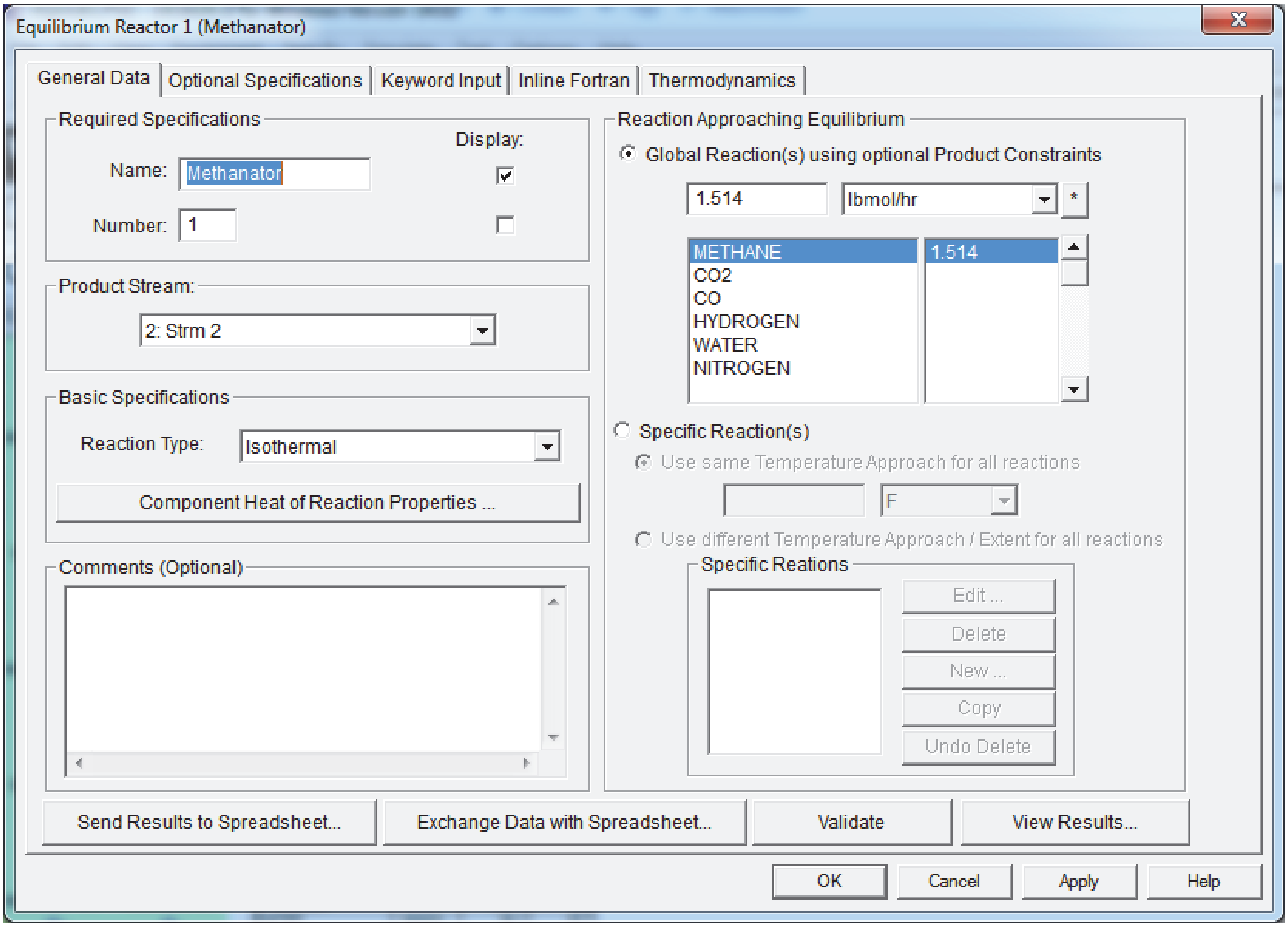


Figure 1: Equilibrium Reactor Dialog (from equlb1.psd)

General Data

Data Items	Description
Name/ Number Display	The name/number associated with the equipment. You can enter a number for the equipment; then choose to display it on the flowsheet.
Product Stream	Open the list and choose the desired stream to use as the product stream.

Equilibrium Reactor Types

Data Items	Description
Adiabatic	Select this command to have the calculation performed adiabatically (constant enthalpy). Default calculation is isothermal (constant temperature).
Isothermal	Select this command to have the calculation performed isothermally (constant temperature).

Reaction(s) Approaching Equilibrium

You can specify the products approach to equilibrium in one of two ways: either GLOBAL or SPECIFIC. GLOBAL and SPECIFIC reaction commands cannot be mixed and they should be used individually.

Global Reaction Commands

Data Items	Description
Product constraint i	Enter the flow of component i in the product stream, where i is the component ID number. This sets the amount of component i wanted in the product.

Specific Reaction Commands

These commands are used to enter specific information about an individual reaction. To use either the Extent of reaction or Temperature approach for Reaction commands, you must first enter a Reaction j command. This command is of the form

Rj = (a1*r1 + a2*r2 + ... = b1*p1 + b2*p2 + ...)

where

a1,a2 are the stoichiometric coefficients for the reactants,

r1,r2	are the component ID numbers for the reactants,
b1,b2	are the stoichiometric coefficients for the products,
p1,p2	are the component ID numbers for the products,
j	is the reaction number.

The stoichiometric coefficients are entered on a molar basis. Fractions are allowed.

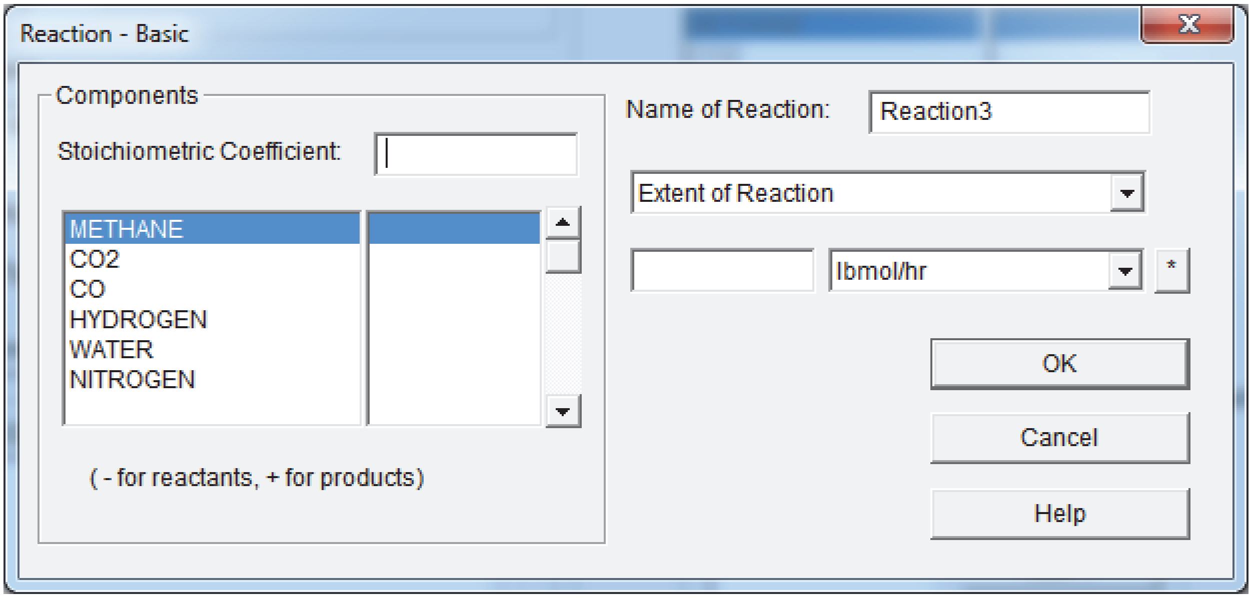


Figure 2: Reaction – Basic (from equlb2.psd)

Specific Reactions

Data Items	Description
Extent of reaction j -OR-	Enter the extent of reaction for reaction j, where j is the reaction number.
Temperature approach for Reaction	Enter the approach to equilibrium temperature for reaction j, where j is the reaction number.

NOTES:

1. Although reactions are described with stoichiometric coefficients on a molar basis, the reactions are not necessarily equimolar. Reactions will always be in mass balance. Having entered the Rj command, enter one of the following two commands.
2. For specific adiabatic reactions using equilibrium reactor, please use negligible amount of CO2 (1E-05) in the feed stream and specify CO2 as an inert substance using INErt=49 command.
3. All individual reactions should be numbered and defined in ascending order (ex. R1, R2, R3 and NOT R2,R1, R3 or 3,R2,R1). A total of 24 reactions are allowed per reactor module.

Defining Heat and Entropy of Formation Data

Reactions may be defined as continuing to equilibrium or approaching equilibrium. For either of the above two cases, heat of formation and entropy of formation data must be supplied. For components with ID numbers from 1 - 98 these data are stored on the pure component data base and automatically accessed. For components with ID numbers greater than 98 you must supply the data at 25 C using the following commands in the General section of the input file (or under Specify...Component...Component Heat of Reaction tab on the Thermodynamic and Transport Methods dialog):

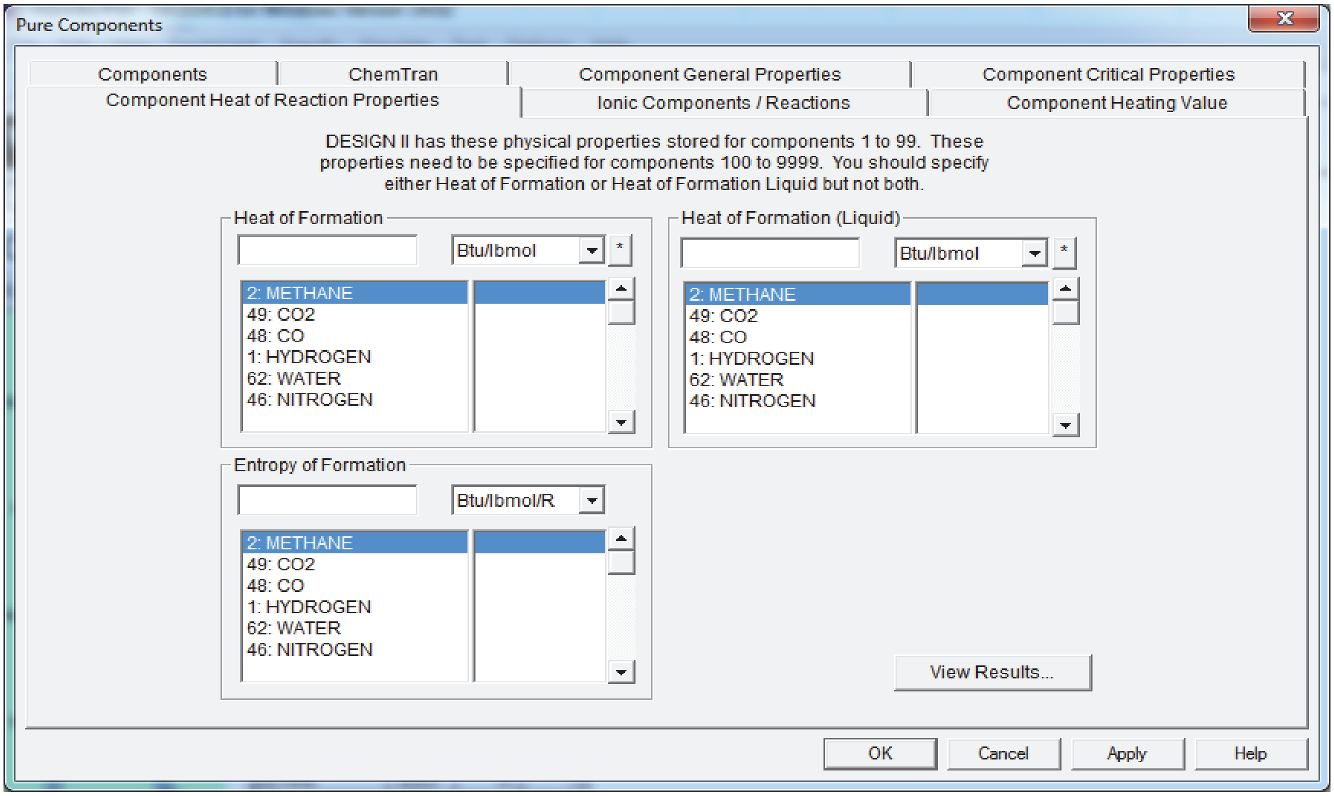


Figure 3: Component Heat of Reaction Properties (from equlb2.psd)

Data Items	Description
Heat of Formation i	Enter the heat of formation where i is the component ID number.
Entropy of Formation i	Enter the entropy of formation where i is the component ID number.

Optional Specifications

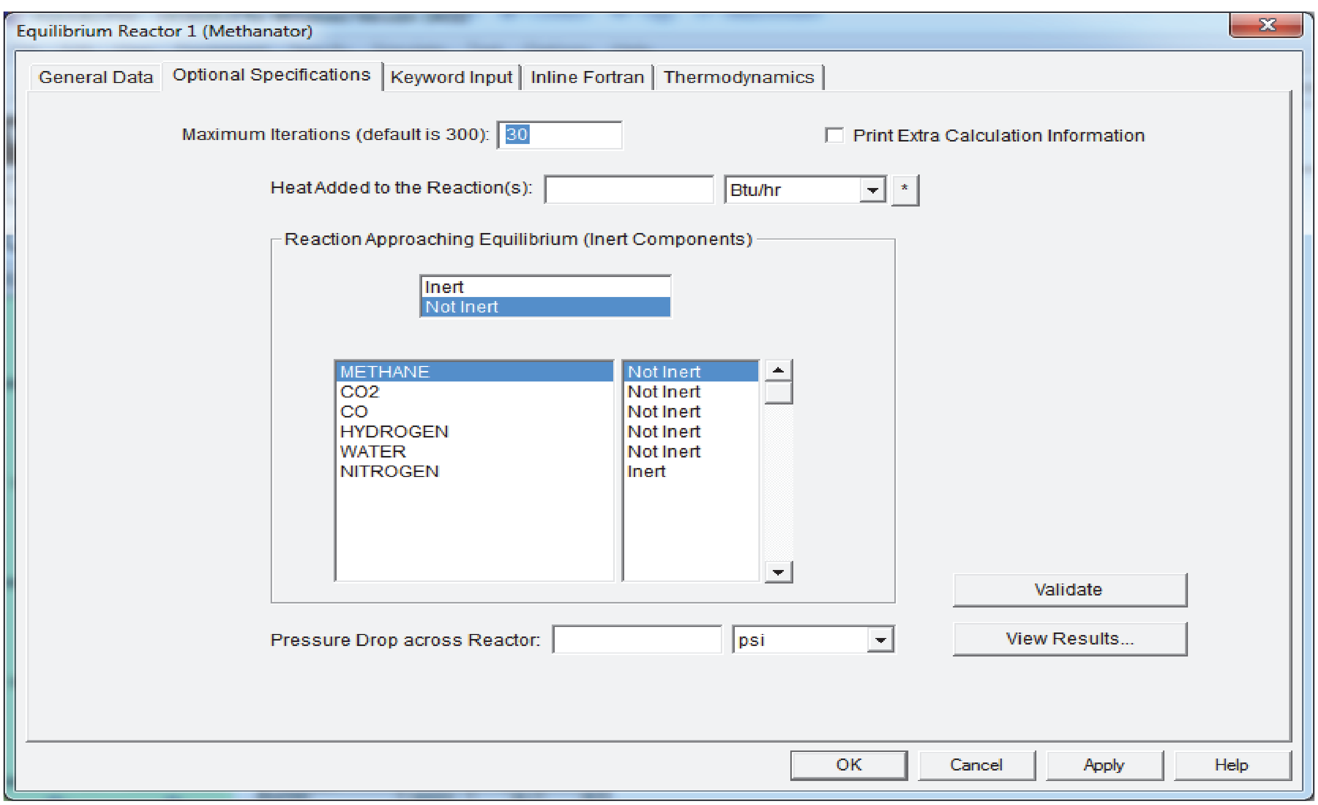


Figure 4: Optional Parameters (from equlb2.psd)

The following commands are optional. They may be used independent of the type of reaction defined.

Data Items	Description
Maximum Iterations	Sets the number of iterations performed. Default is 30.
Heat Added to the Reaction(s)	Can be used to add more heat (or cool) to the reactions.
Inert	Enter the ID numbers of the components which are not involved in the reaction(s). These components are inert; they do not participate in the reactions, but do participate in the enthalpy balances.
Pressure Drop across Reactor	Enter the pressure drop and select a unit.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Equilibrium Reactor Examples

There are several sample flowsheets in “Chapter 19: Equilibrium Reactor Samples - c:\designii\samples\equipmnt\equireac” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.14: Expander

General

The Expander module simulates a gas or liquid turboexpander. The work of expansion is calculated from the isentropic expansion as corrected by the adiabatic efficiency.

Details

The Expander module performs an isentropic expansion of liquid, vapor, or two-phase streams to a specified outlet pressure. Actual work is then calculated from the ideal work and the efficiency. Finally, the temperature and phase distribution are calculated using the actual work. The calculated work is stored as a positive number. You may use the calculated work from an Expander to drive a compressor with either the Controller module or Inline Fortran.

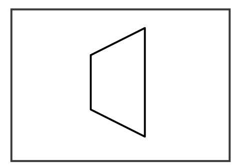
Input streams are adiabatically mixed to the lowest feed pressure before expansion. If you use two outlet streams, vapor will go into first outlet and liquid into the second outlet stream.

NOTE: If the inlet streams are liquid, only one outlet stream should be used.

Please see the online **DESIGN II** Help topic ***Equipment/Expander*** or the **DESIGN II** Reference ***Guide Chapter 3.13: Expander*** for more details.

Symbols

The Expander unit module has one symbol:



The Expander module requires that one inlet and one outlet stream be connected to the module. You can have multiple inlet streams and up to two outlet streams connected to this module.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

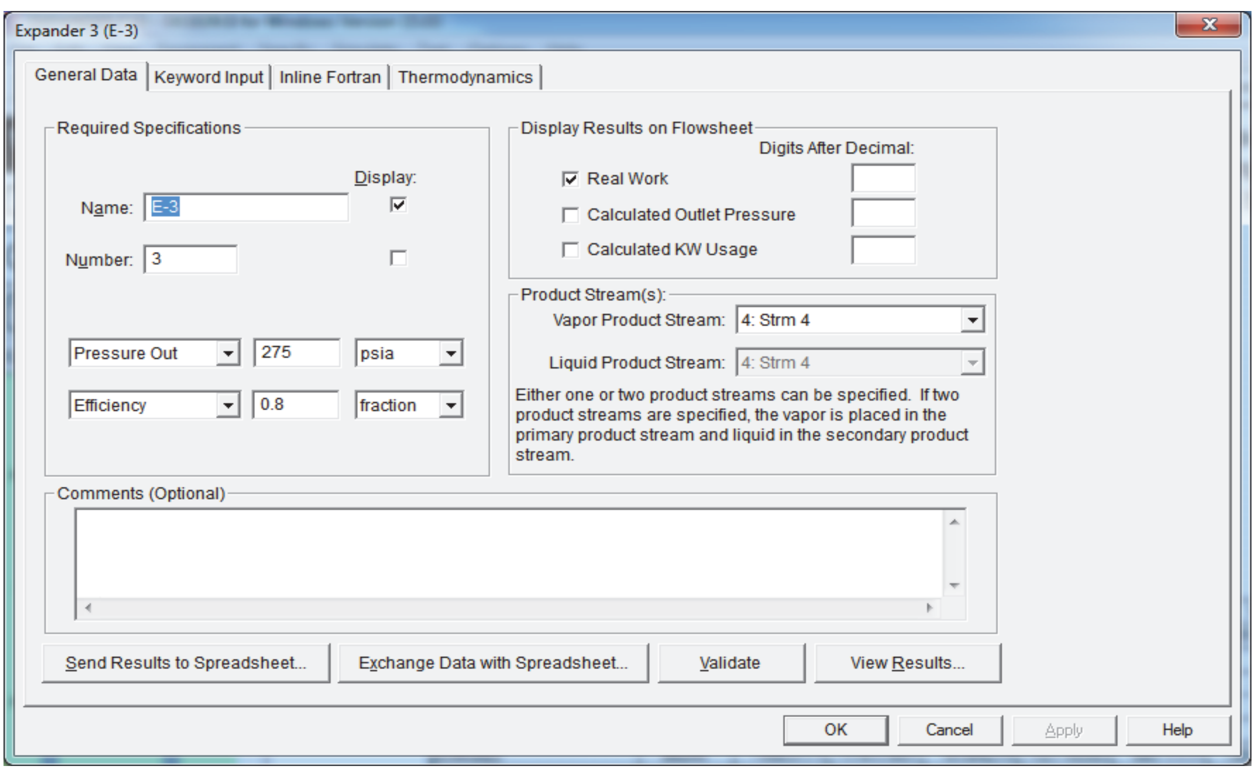


Figure 1: Expander Dialog (from expander.psd)

General Data

This tab is used to enter the basic specifications for the Expander Module.

Data Item	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Display Results on Flowsheet	You can optionally display the following results on the flowsheet: Real Work Calculated Outlet Pressure Calculated KW usage If you select to display one or more of these results, enter the number of digits to display to the right of the decimal.
Pressure Out	Specified pressure at outlet
Delta Pressure	Desired pressure change or delta pressure. The default is twice the suction pressure.
Isentropic Efficiency	Ratio of isentropic work to actual work. The default is 0.72.
Product Streams	Select a Vapor Product and/or Liquid Product Stream from the drop down list; if two product streams are specified, the vapor is placed in the primary product stream and liquid in the secondary product stream.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II Reference Guide.

Optional Specifications: Keyword Input

Click **Keyword Input**. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Expander Examples

There are several sample flowsheets in “Chapter 20: Expander Samples - c:\designii\samples\equipmnt\expander” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.15: Expander Compressor

General

The Expander Compressor module is a utility module that passes or uses the calculated work by the expander to the compressor as the maximum available work.

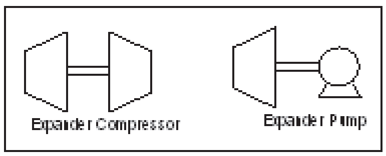
Details

The expander part of the module is similar to the stand alone EXPander module that performs an isentropic expansion of liquid, vapor, or two-phase streams to a specified outlet pressure; actual work is then calculated from the ideal work and the user-supplied efficiency. The stored CALculated WORk is a positive number and will be automatically used by the compressor portion of the module as the maximum available work.

Only two input streams and two output streams can be assigned for the combined module by the user. If the isentropic calculation is specified, a rigorous mixed-phase calculation is performed. If the isentropic calculation is not specified, the liquid is compressed as if it were a vapor. If the compressor feed vapor fraction is less than 0.9, the module shifts to a pump calculation.

Symbols

The Expander Compressor and Expander Pump have one symbol each but share the data items.



You must connect two inlet streams and two outlet streams to the Expander Compressor/Pump module.

Properties

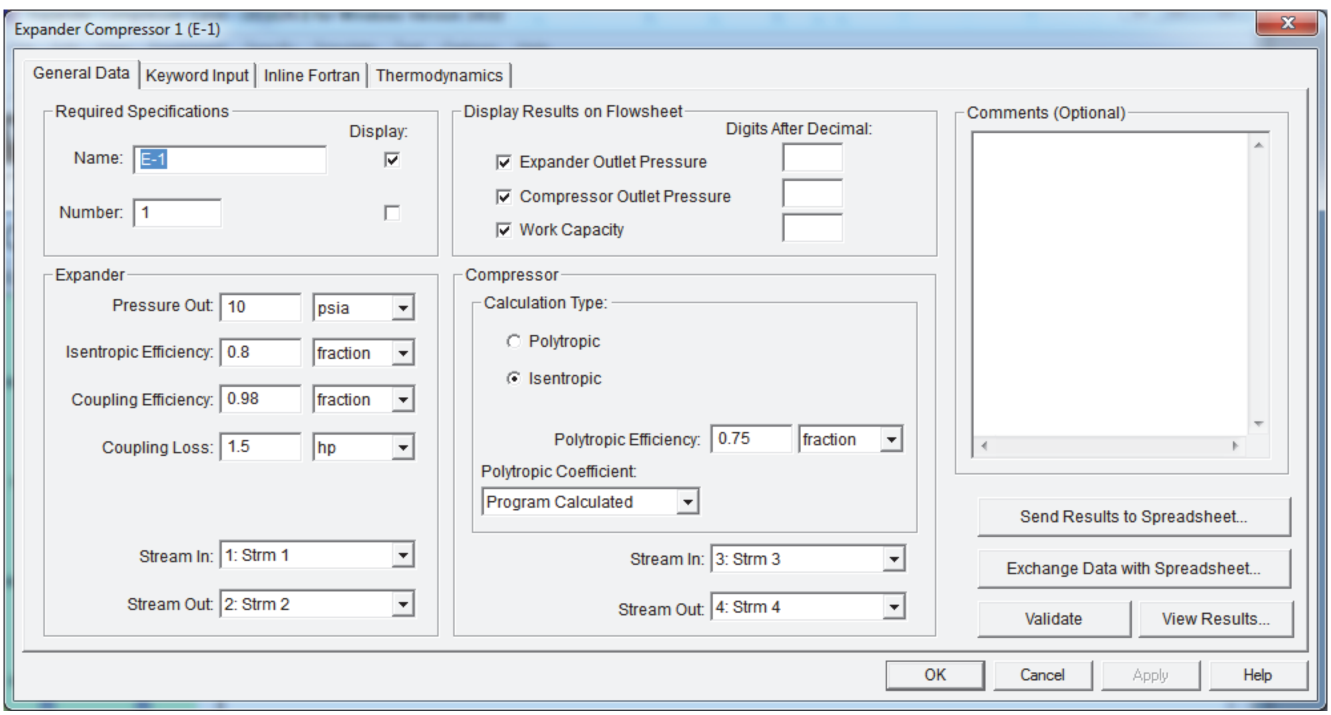


Figure 1: Expander Compressor Dialog (from expander compressor1.psd)

General Data

Data Items	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Display Results on	You can optionally display the following results on the flowsheet:Flowsheet Expander Outlet Pressure Compressor Outlet Pressure Work Capacity If you select to display one or more of these results, enter the number of digits to display to the right of the decimal.
Expander	
Pressure Out	Enter desired discharge pressure for the expander compressor module. Default is one half of suction pressure.
Isentropic Efficiency Expander	Enter the isentropic efficiency of the expander portion of the EXPander COMpressor module. Default is 0.72.
Isentropic Efficiency Compressor	Enter the isentropic efficiency of the compressor/pump portion of the EXPander COMpressor module. Default is 0.65.
Coupling Efficiency	Enter the coupling efficiency on the work produced by expander to be used by compressor. Default is 1.0.
Coupling Loss	Enter the coupling loss due to lost mechanical work by the compressor. Default is 0.0 HP.
Stream In	Select the name of the stream to use as the inlet for the expander.
Stream Out	Select the name of the stream to use as the outlet for the expander.
Compressor	
Calculation Type	Select one of the following: Polytropic Isentropic To simulate a centrifugal compressor when you know the Cp/Cv ratio and polytropic efficiency, select polytropic. If you have manufacturer's curves for an existing centrifugal compressor, you should consider using polytropic. The temperature for the polytropic coefficient of the compressor case is calculated from the enthalpy difference due to the pressure change and work added to the system.
Polytropic Efficiency/ Isentropic Efficiency	Enter the polytropic or isentropic efficiency, depending on your calculation type selection. The default for polytropic efficiency is 0.72 fraction. The default for isentropic efficiency is 0.65 fraction. Then select a unit.
Polytropic Coefficient	If you selected Polytropic as the calculation type; then select one of the following: Program Calculated: This is the default option. Polytropic Coeff: Enter the polytropic coefficient [(n-1)/n]. The default for the dimensionless coefficient is 0.22 (with n = 1.282).
Stream In	Select the name of the stream to use as the inlet for the compressor.
Stream Out	Select the name of the stream to use as the outlet for the compressor.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module. Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II Reference Guide.

Expander Compressor: Example

There are several sample flowsheets in “Chapter 21: Expander Samples - c:\designii\samples\equipmnt\expcom” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.16: Fired Heater

General

The Fired Heater module calculates the fuel consumption and actual heater duty when a process stream is heated to a given temperature.

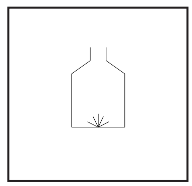
Details

The heating value of the fuel and the heater efficiency can be specified along with the rated duty of the heater. An essentially unlimited number of input streams can be coded; they will be adiabatically mixed to the lowest feed pressure. Either one or two output streams can be coded. If two output streams are coded, the vapor is placed in the first output and liquid in the second output.

Please see the online **DESIGN II** Help topic ***Equipment/Fired Heater*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 15: Fired Heater*** for more details.

Symbols

The Fired Heater unit module has one symbol:



The Fired Heater module requires that at least one inlet stream and at least one outlet stream be connected to the module.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

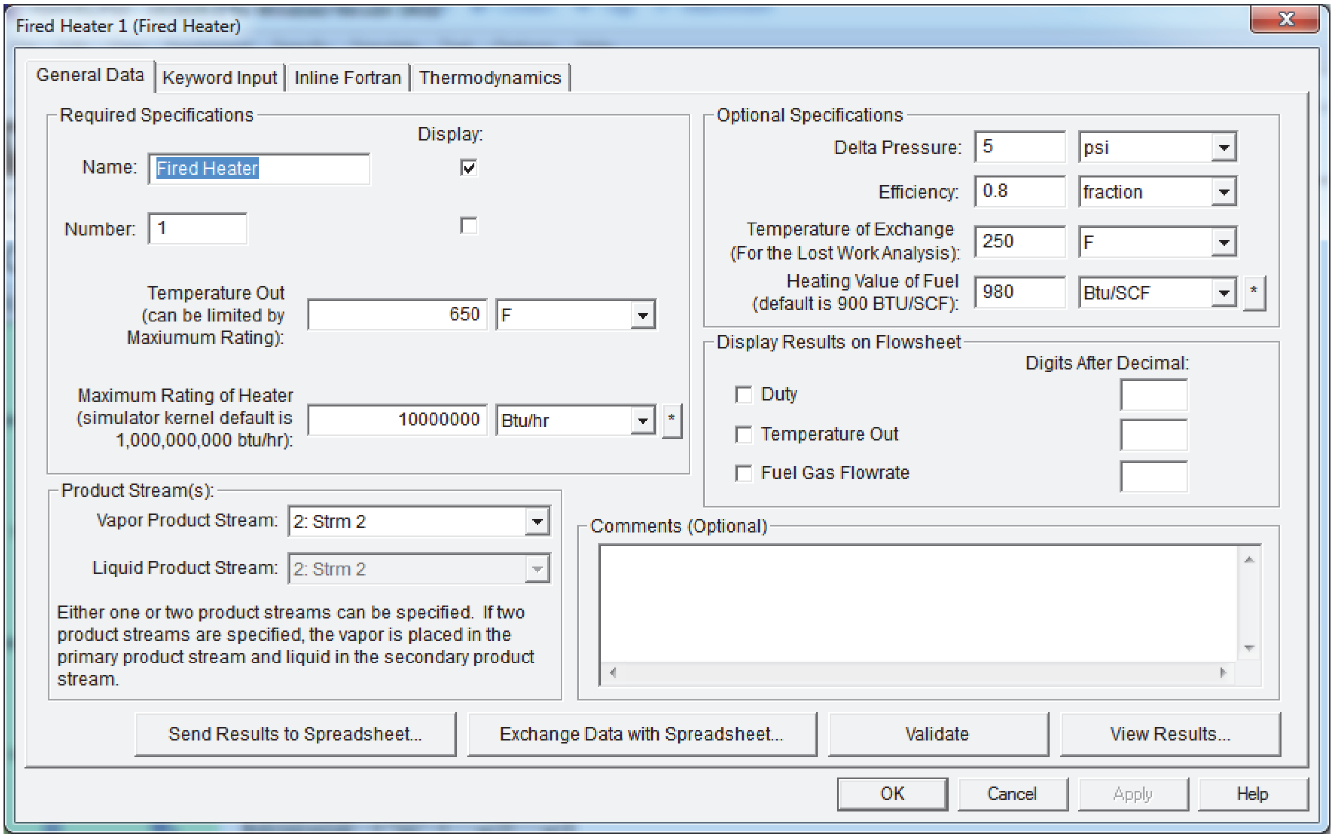


Figure 1: Fired Heater Dialog (from fdhtr1.psd)

General Data

Data Items	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Temperature Out	Enter the desired temperature of the process stream.
Maximum Rating of Heater	Enter the rated duty to be used as the heating limit. Outlet temperature will be reduced if this limit is exceeded. Default is 1 x 10**9 BTU/HR (2.93 x10**5 KJ/SEC).
Display Results on	You can optionally display the following results on the flowsheet:Flowsheet Duty Temperature Out Fuel Gas Flowrate If you select to display one or more of these results, enter the number of digits to display to the right of the decimal.
Optional Specifications	
Delta Pressure	Enter the pressure drop.
Efficiency	Enter the fuel efficiency. Default is 0.75.
Temperature of Exchange	Enter the exchange temperature for the LOST WORK analysis. Default is 250F.
Heating Value of Fuel	Enter the heating value of the fuel. Default is 900 BTU/SCF.
Product Streams	Select a Vapor Product and/or Liquid Product Stream from the drop down list; if two product streams are specified, the vapor is placed in the primary product stream and liquid in the secondary product stream.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the "C-*" before it and enter the units in parenthesis and any values after the =. Any line beginning with "*" will not be written to the DESIGN II input file. However any line with a "C-" (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II Reference Guide.

Fired Heater Example

There are several sample flowsheets in "Chapter 22: Fired Heater Samples - c:\designii\samples\equipmnt\firedhea" of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.17: Flash

General

The Flash module separates a vapor and liquid phase of a single two-phase stream. Three outlet streams can be used if water is in your list of chemical components and has been specified as immiscible.

Details

The flash module can separate the vapor and liquid phase of a single two-phase stream. Up to three outlet streams and as many inlet streams as the number of snap points permit can be connected to the module. If you specify three outlet, streams you must enter water in your list of components using the Components dialog under the Specify Menu. You must also specify water as immiscible using the Advanced Thermo under the same menu.

The first outlet stream will contain the vapor phase; the second outlet stream will contain the hydrocarbon liquid phase plus soluble water; the third outlet stream will contain "free" water plus soluble hydrocarbons. For a three-phase hydrocarbon-water system, you can also specify entrainment in the two liquid phases. A heat loss or gain can be specified for adiabatic calculations. If heat change is not specified, the phase separation will be isothermal.

A variety of calculations can be performed with the flash module. In addition to the isothermal and adiabatic flash you can specify an isentropic flash. You may also specify that the calculations be done at the outlet conditions rather than the inlet conditions. This requires specification of either the outlet temperature or pressure. Bubble point and dew point conditions may also be specified.

You can request sizing calculations for these single stage separators on your flowsheet by entering a few extra commands in the optional Keyword Input dialog. These calculations are intended for preliminary design work only and should not be used for detailed vessel design to A.S.M.E. code requirements. Simplifying assumptions have been made to facilitate preliminary sizing. DESIGN II can size any vertical or horizontal liquid-vapor separators such as flash drums, reflux drums, accumulators, water knockout drums and inlet gas or compressor inlet separators.

Different types of separators can be sized in a single run using sizing commands available in either the Flash or Valve module. Physical properties of the vapor and liquid phases are obtained automatically from the program. These properties correspond to the vessel outlet conditions. An appropriate selection of K-value and density options should be made to obtain best results.

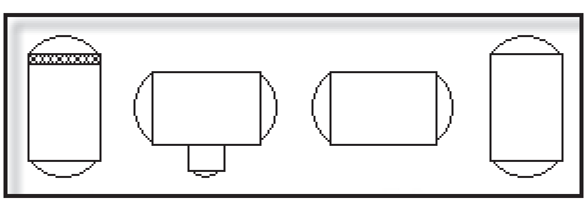
DESIGN II gives you a wide choice of sizing specifications and uses reasonable default values if you do not specify any. Vessel diameter, length and thickness are calculated, and total weight of steel is estimated using a percentage allowance for supports and fittings. A scaled sketch is given for vertical vessels indicating the relative dimensions, liquid level and nozzle positions.

An unlimited number of input streams may be coded. When no specification is given and more than one input stream is coded, DESIGN II performs an adiabatic flash at the lowest feed pressure. However, if only one feed stream is coded and no specification is given; then DESIGN II performs an isothermal flash at the feed temperature and pressure.

Please see the online **DESIGN II** Help, topic ***Equipment/Flash*** or the **DESIGN II Unit Module** Reference Guide ***Chapter 16: Flash*** for more details.

Symbols

The Flash unit module has four symbols: Flash 1 (vertical separation vessel), Flash 2 (horizontal separation vessel with boot), Flash 3 (horizontal separation vessel without boot), and Flash 4 (vertical separation vessel withou a mist eliminator).



The Flash module requires that at least one inlet and two outlet streams be connected to the module. Depending on the number of snap points, multiple inlet streams and up to three outlet streams may be connected to this module.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

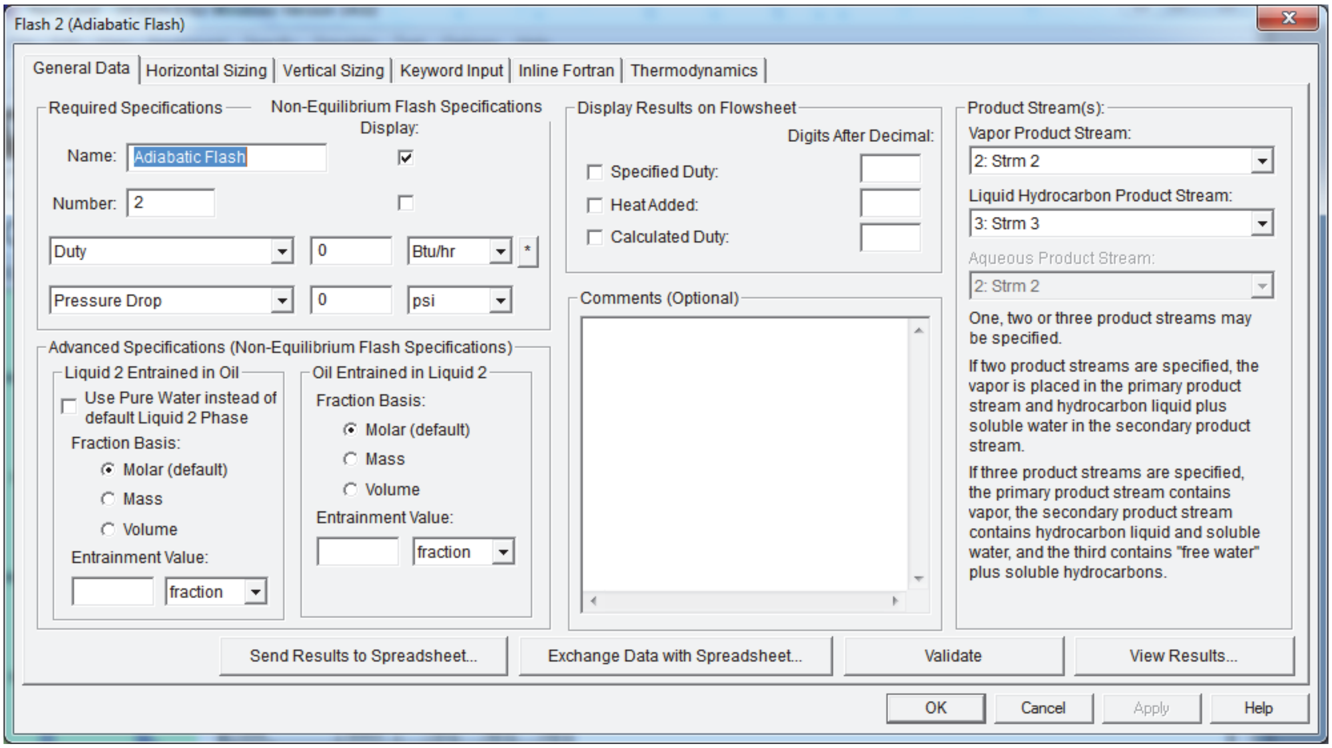


Figure 1: Flash Dialog (from flash1.psd)

General Data

This dialog provides a temperature and pressure combo box for entering the basic specifications for the Flash Module. Temperature specifications are made in the top combo box and pressure specifications in the bottom combo box.

Data Item	Description
Temperature Specs	
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Display Results on Flowsheet	You can optionally display the following results on the flowsheet: Specified Duty Heat Added Calculated Duty If you select to display one or more of these results, enter the number of digits to display to the right of the decimal.
Temperature	
Duty	Enter the heat change (negative for a loss).
Temperature	Enter the outlet temperature for the flash.
Tem from Feed	Leave the outlet temperature the same as the inlet temperature and specify the outlet pressure.
Bubble Pt. Tem	Perform an isothermal flash for outlet conditions at the bubble point and a specified pressure.
Dew Pt. Tem	Perform an isothermal flash for outlet conditions at the dew point and a specified pressure.
Water Dew Tem	Calculate the temperature at the dew point of water. Water must be selected for the components list.
Isentropic	Perform an isentropic flash for a specified pressure.
Liq Frac (Mol)	Calculate the temperature to achieve the liquid fraction entered (in molar units) at a specified pressure.

Liq Frac (Mas)	Calculate the temperature to achieve the liquid fraction entered (in mass units) at a specified pressure.

Pressure

Pressure Drop	Enter the pressure drop for the flash calculation.
Pressure	Enter the outlet pressure for the flash calculation.
Pressure from Feed	Specify that the outlet pressure will be the same as the inlet pressure.
Bubble Pt. Pres	Perform an isothermal flash for outlet conditions at the bubble point and a specified temperature.
Dew Pt. Pres	Perform an isothermal flash for outlet conditions at the dew point and a specified temperature.
Water Dew Pres	Calculate the pressure at the dew point of water. Water must be selected for the components list.
Isentropic	Perform an isentropic flash for a specified temperature.
Liq Frac (Mol)	Calculate the pressure to achieve the liquid fraction entered (in molar units) at a specified temperature.
Liq Frac (Mas)	Calculate the pressure to achieve the liquid fraction entered (in mass units) at a specified temperature.

Advanced Specifications

Water Entrained in Oil	For Fraction Basis, select either: Molar Mass Volume Entrainment Value: Enter the fraction for water entrained in oil.
Oil Entrained in Water	For Fraction Basis, select either: Molar Mass Volume Entrainment Value: Enter the fraction for oil entrained in water.
Product Streams	Select a Vapor Product, Liquid Hydrocarbon Product Stream, and/or Aqueous Product Stream from the drop down list. If two product streams are specified, the vapor is placed in the primary product stream and liquid in the secondary product stream. If three product streams are specified, the vapor is placed in the primary product stream, hydrocarbon liquid and soluble water in the secondary product stream, and “free water” plus soluble hydrocarbons in the third product stream.

Optional Specifications: Horizontal Sizing

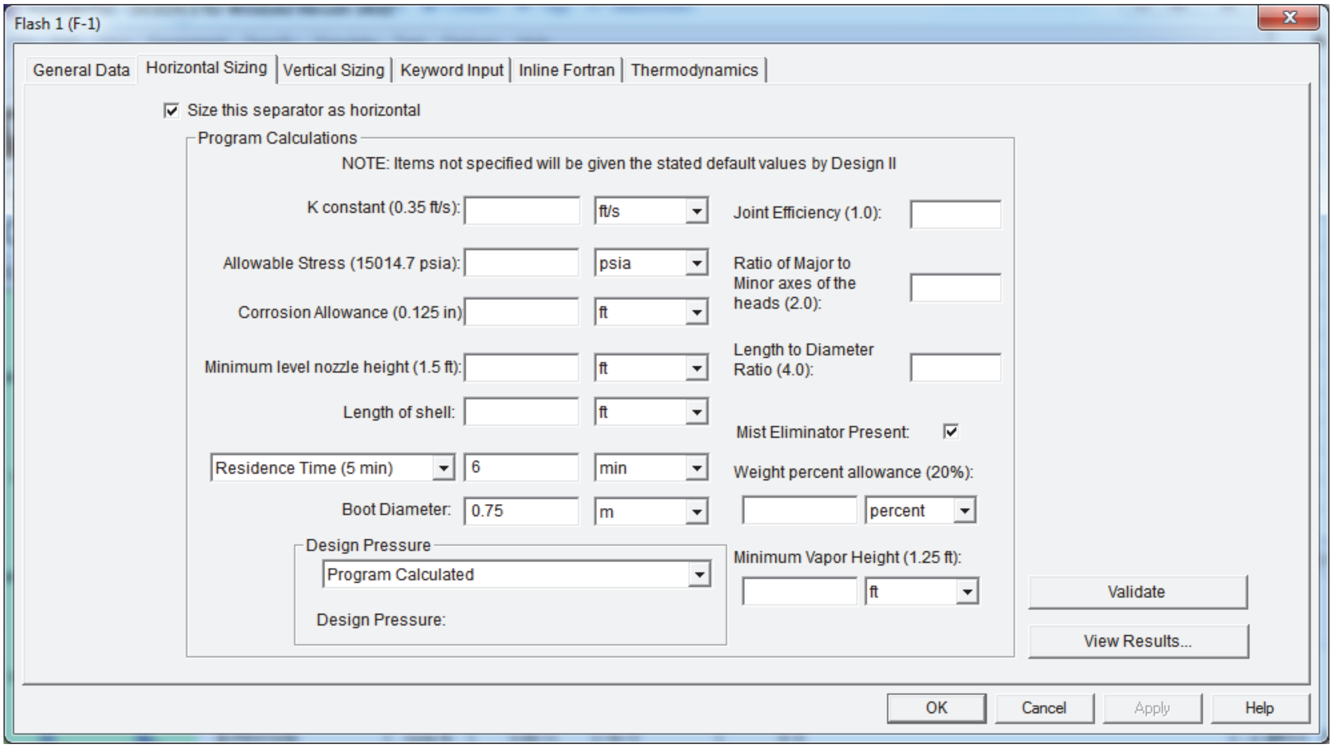


Figure 2: Horizontal Sizing (from flash8.psd)

Data Item	Description
K Constant	Enter the K constant (velocity). Default is 0.35.
Joint Efficiency	Enter the joint efficiency. Default is 1.0.
Allowable Stress	Enter the allowable stress. Default is 15014.7 PSIA.
Ratio of Major to Minor Axes of the heads	Enter the ratio of major to minor axes for the heads. Default is 2.0.
Corrosion allowance	Enter the corrosion allowance. Default is 0.125 inches.
Minimum level nozzle height	Enter the minimum level nozzle height from tangent line to high liquid level. Default is 1.5 feet. This command can be used to ensure that your vessel has sufficient liquid level for proper placement of level control nozzles.
Length to diameter ratio	Enter the ratio of length to diameter. Default is 4.0. This ratio is used to calculate the length of the vessel.
Length of Shell	Enter the length of shell. There is no default value for this command.
Mist Eliminator	Enter this command to specify that the vessel has a mist eliminator. Default option.
Residence Time/Inside Diameter	Residence Time: Enter the residence time. Default is 5 minutes. **NOTE:** The default unit for time is hours. The residence time will be used to determine the normal minimum liquid level. This should include any additional settling time needed for separation of two liquid phases. Inside Diameter: Enter the diameter and select a unit. The residence time will be calculated.
Boot Diameter	Enter the boot diameter if you want to separate two liquid phases in the vessel. The second liquid phase will be contained in the boot. The program will calculate the size of the horizontal vessel required for the vapor and first liquid phase and the length of the boot for the second liquid phase. There is no default value.
Weight Percent Allowance	Enter the weight percent allowance. Default is 20.0
Design Pressure	Enter the design pressure. Default is 10 percent higher than the operating pressure.
Minimum Vapor Height	Enter the minimum vapor height. Default is 1.25 feet.

Optional Specifications: Vertical Sizing

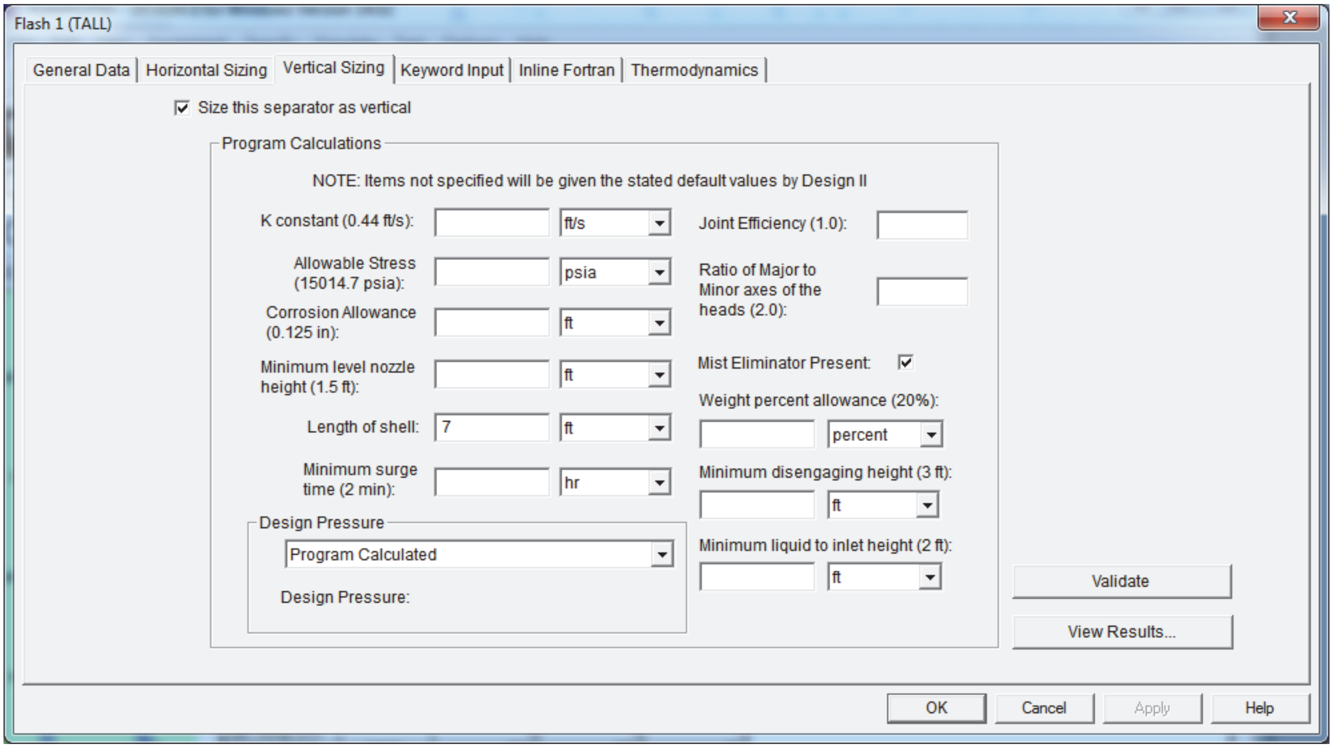


Figure 3: Vertical Sizing (from flash5.psd)

The sizing calculation is performed if you specify either:

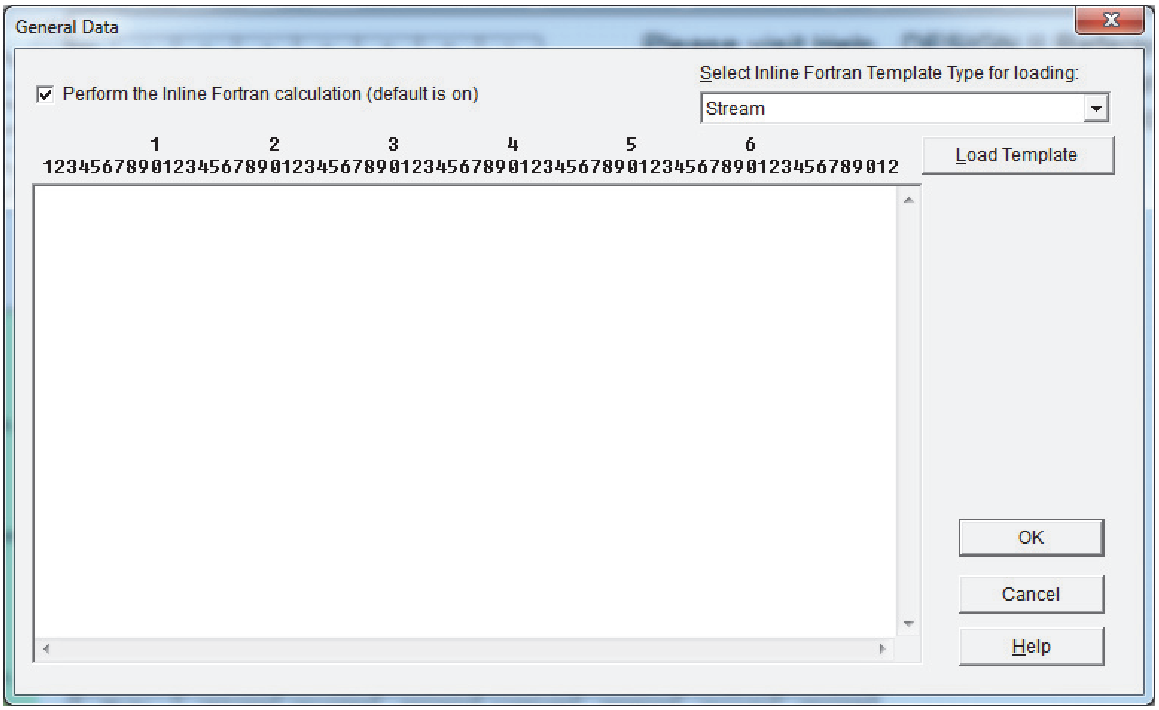
Data Item	Description
K Constant	Enter the K constant (velocity). Default is 0.35.
Joint Efficiency	Enter the joint efficiency. Default is 1.0.
Allowable Stress	Enter the allowable stress. Default is 15014.7 PSIA.
Ratio of Major to Minor Axes of the heads	Enter the ratio of major to minor axes for the heads. Default is 2.0.
Corrosion allowance	Enter the corrosion allowance. Default is 0.125 inches.
Minimum level nozzle height	Enter the minimum level nozzle height from tangent line to high liquid level. Default is 1.5 feet. This command can be used to ensure that your vessel has sufficient liquid level for proper placement of level control nozzles.
Mist Eliminator	Enter this command to specify that the vessel has a mist eliminator. Default option.
Length of Shell	Enter the length of shell. There is no default value for this command.
Weight Percent Allowance	Enter the weight percent allowance. Default is 20.0
Minimum surge time	Enter the minimum surge time. This will be used to determine the high liquid level measured from the tangent line of the vessel. Default is 2 minutes.
Minimum disengaging height	Enter the minimum disengaging height. Default is 3 feet. **NOTE:** The lower limit for this specification is the diameter of the vessel.
Design pressure	Enter the design pressure. Default is 10 percent higher than the operating pressure.
Minimum Liquid to inlet height	Enter the minimum distance between the inlet nozzle and the high liquid level. Default is 2 feet. **NOTE:** The actual distance between the inlet nozzle and the high liquid level is at least one half the vessel diameter. It will be increased, if necessary, to satisfy both the minimum liquid to inlet height and the total vessel length.

Optional Specifications: Keyword Input

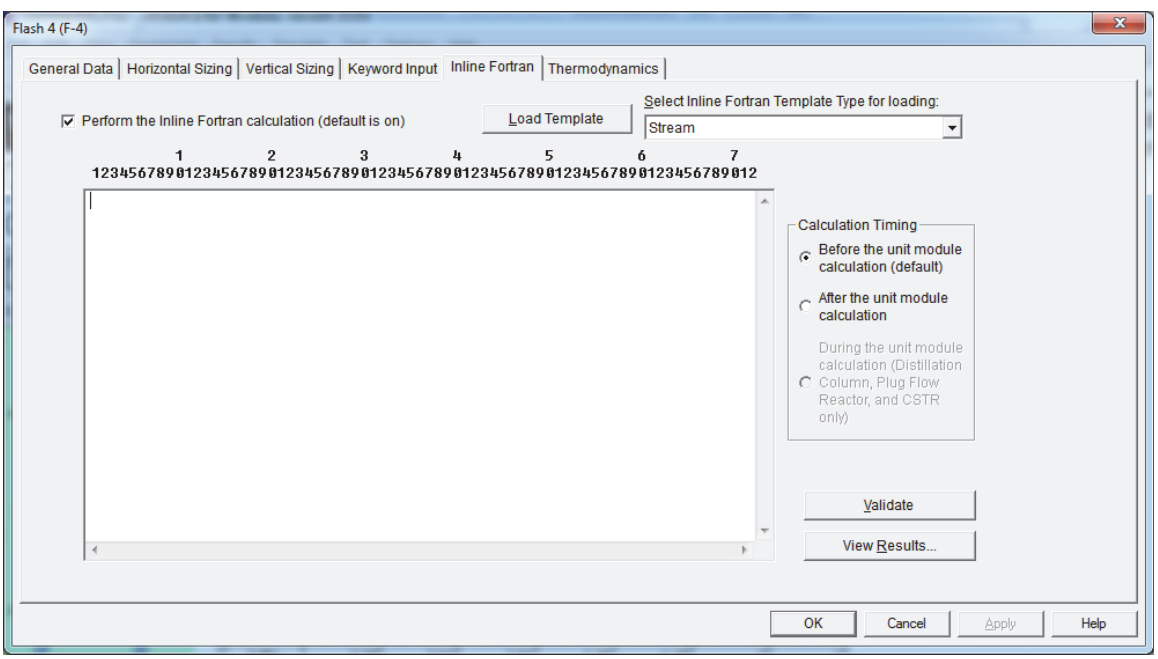
Click the **Keyword Input** tab. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the "C-*" before it and enter the units in parenthesis and any values after the =. Any line beginning with "*" will not be written to the DESIGN II input file. However any line with a "C-" (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.



The following dialog must be included for ALL module sections to replace the current Inline Fortran dialog. Flash module is shown below (need to replace for ALL modules)



Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Flash Examples

There are several sample flowsheets in “Chapter 23: Flash Samples - c:\designii\samples\equipmnt\flash” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.18: Flow Meter

General

The Flow Meter module is applicable to the orifice type flow meter and allows the engineer to calculate the size of the orifice, flow rate of the stream, or the pressure drop across the orifice. Additionally, this model can provide tables and parametric plots of flow rate verses pressure drop. The flow condition can be single or two phase in the subsonic region and vapor phase only in the sonic flow region.

The model allows square-edge, quadrant, and conical type orifice plates. The International Standard Organization method and the American Gas Association methods are available to correlate discharge coefficient and expansion factor for square-edged orifice. This model can accommodate flange, corner, radius, or pipe orifice taps. See Flow Meter: Range of Application for orifice and tap types.

Note: The input order in this module is not totally free-format. Commands should be entered in the order that they are organized in the DESIGN II Keyword Commands section.

Details

The following table lists the range of application for the available orifice plate and tap types and the available methods for calculating the discharge coefficient and expansion factor.

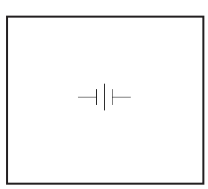
Method	Tap Type	Pipe Diameter	Beta-Ratio or Diameter of Orifice	Reynold's Number (Rd* or RD*)
ISO (square-edged	flange, corner, radius	2 inch< D< 36 inch	$0.2 < \beta < 0.75$	2,000 - 10,000,000 (RD)
orifice)	pipe	2 inch< D< 36 inch	$0.2 < \beta < 0.70$	10,000 - 10,000,000 (RD)
AGA	flange	D > 1.6 inch	$0.1 < \beta < 0.75$	---
(square-edged orifice)	pipe	D > 2 inch	$0.2 < \beta < 0.67$	---
Quadrant	flange, corner, radius	D > 2 inch	0.225 0.4 0.5 0.6 0.63	5,000 - 60,000 (Rd) 5,000 - 150,000 (Rd) 4,000 - 200,000 (Rd) 3,000 - 150,000 (Rd) 3,000 - 105,000 (Rd)
Conical	n/a	n/a	D > 0.25 in. $\beta < 0.316$	250 - 200000 (Rd)

* **RD** is the Reynold's No. based on the pipe diameter D, **Rd** is the Reynold's No. based on the orifice diameter d.

Please see the online **DESIGN II** Help topic ***Equipment/Flow Meter*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 17: Flow Meter*** for more details.

Symbols

The Flow Meter unit module has one symbol:



The Flow Meter module requires that one inlet stream and one outlet stream be connected to the module.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

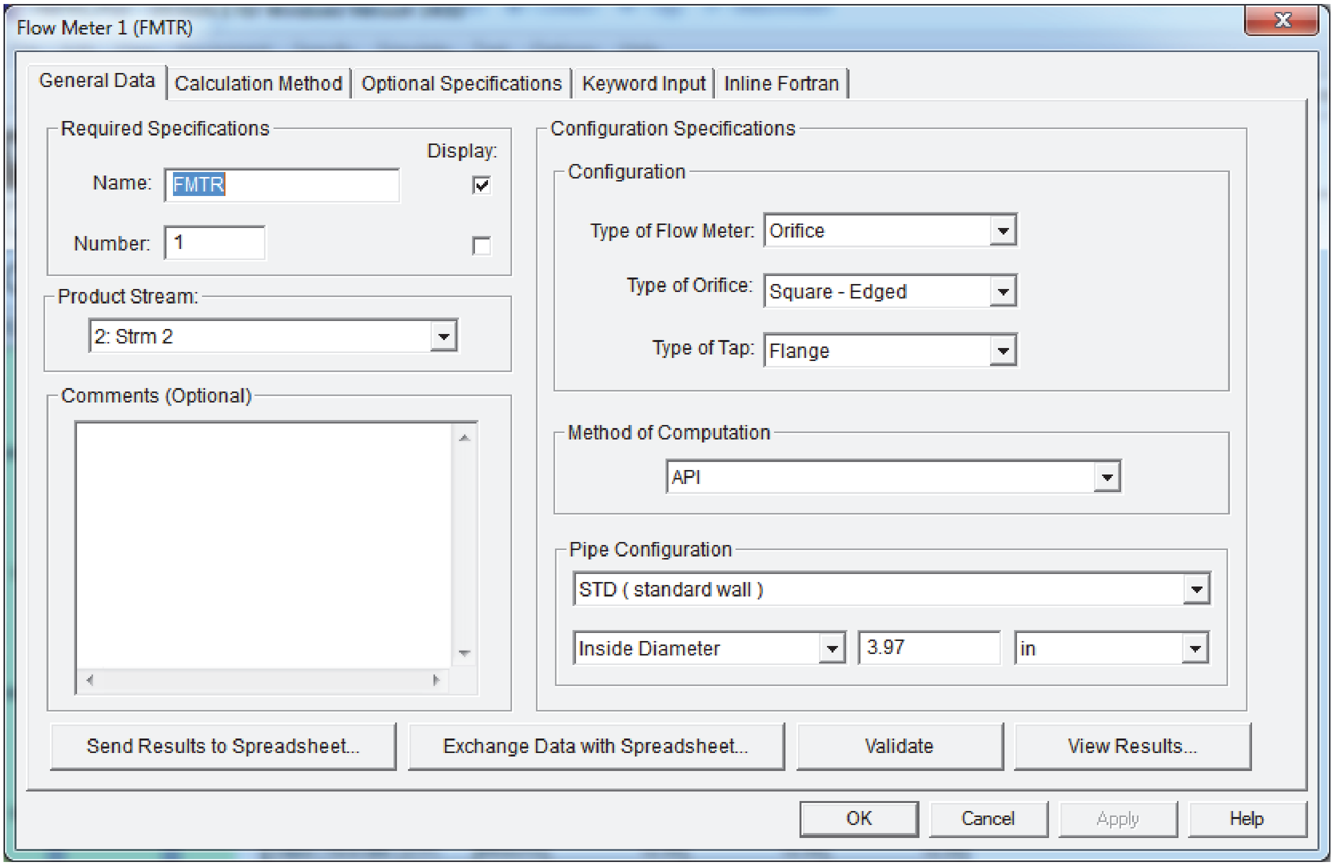


Figure 1: Flow Meter Dialog (from fmtr1.psd)

General Data

Data Item	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Product Stream	Open the drop down list and select a stream to use as the product stream.
Type of Flow Meter	Orifice is the only choice.
Type of orifice	Enter your choice of orifice type. Options are: SQUared-edged (default), QUAdrant, CONical
Type of Tap	Enter your choice of tap type. Options are: FLAnge (default), CORner, RADius, PIPe
Method of computation	Enter the choice of computation method. The options are: ISO available for flange, corner, radius and pipe taps. (default) AGA available for flange and pipe taps only if a square edge orifice is specified. API available for flange and square-edged orifice meters only.

The pipe configuration can be defined by either specifying the nominal diameter and pipe wall thickness code or by specifying the inside diameter alone.

Pipe Wall thickness code	Enter the desired pipe wall thickness code. Options are: STD (standard wall) (default), XS extra strong wall), XXS (double extra strong wall)
Nominal Diameter	Enter the nominal diameter for the pipe. They are consistent with Crane Technical Paper 410.
Inside Diameter	Enter the inside diameter of the pipe.

Required Specifications: Calculation Method

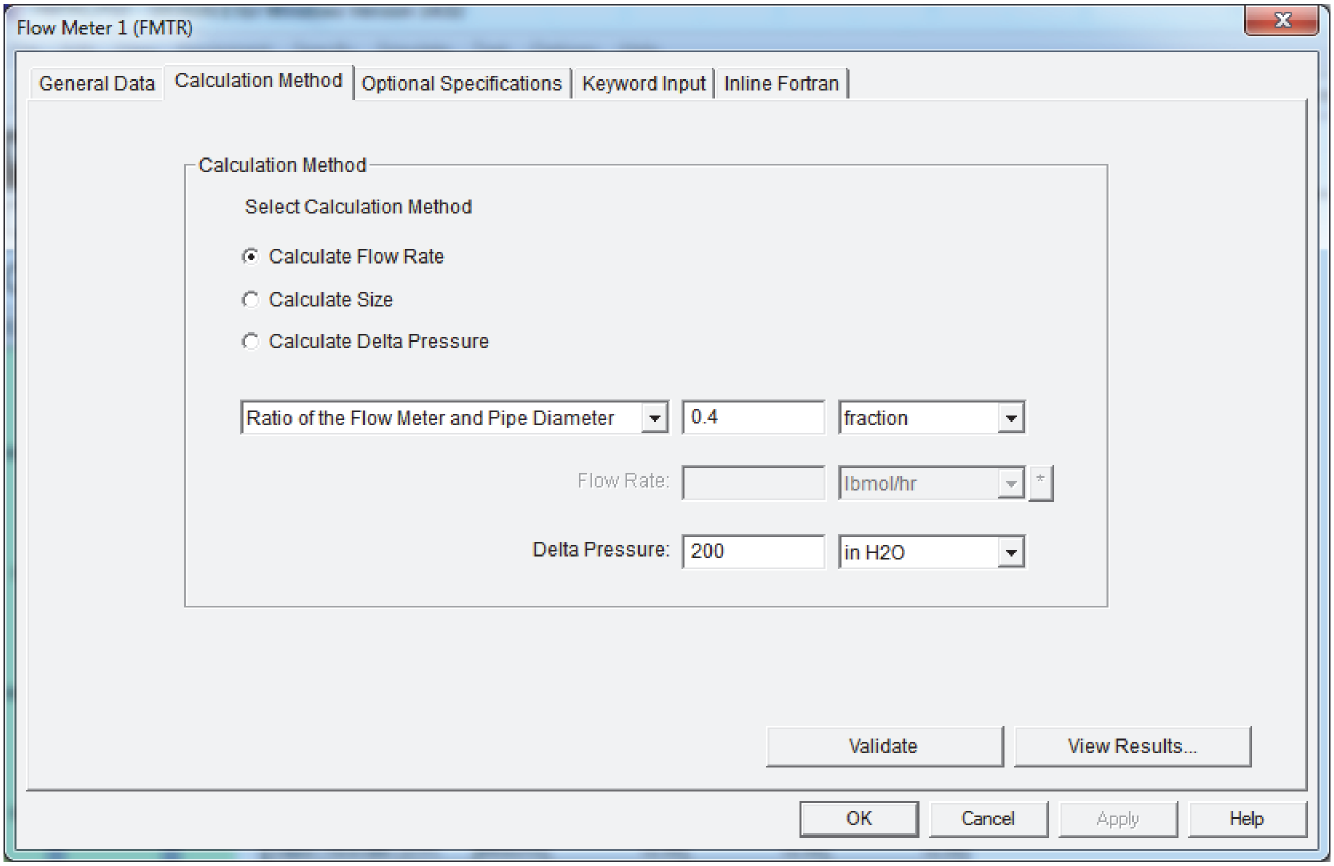


Figure 2: Calculation Method Tab (from fmtr1.psd)

Specify Desired Computation Option

Data Items	Description
Calculate Flow rate	Enter this command to compute the flow rate. The diameter of the flow meter (or ratio) and the delta pressure must be specified.
Calculate Size	Enter this command to compute the diameter of the flow meter. The flow rate and the delta pressure must be specified.
Calculate Delta Pressure	Enter this command to compute the delta pressure of the flow meter. The diameter of the flow meter (or ratio) and the flow rate must be specified.

If you selected Calculate Flowrate or Calculate Delta Pressure, select either:

Diameter of Flow Meter	Enter the diameter and select a unit.
Ratio of the Flow Meter and Pipe Diameter	Enter the ratio and select a unit.

If you selected Calculate Size or Calculate Delta Pressure:

Flowrate	Enter a flowrate and select a unit. To enter base quantity and time units, click the *.

If you selected Calculate Flowrate or Calculate Size:

Delta Pressure	Enter a delta pressure and select a unit.

Required Specifications: Calculation Method

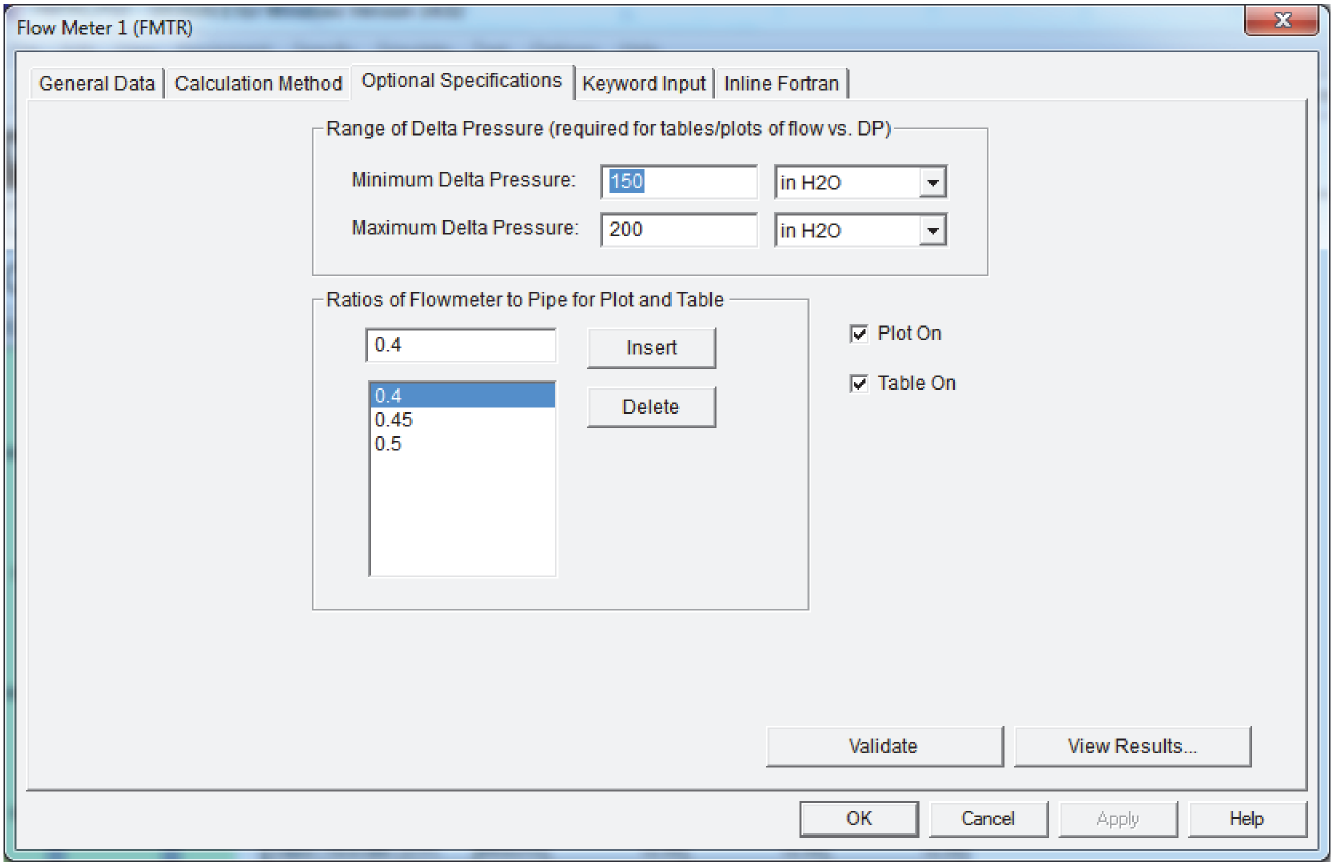


Figure 3: Optional Specifications Tab (from fmtr1.psd)

Specify Desired Computation Option

Data Items	Description
Range of Delta Pressure	This is required to display table plots of flow versus Delta Pressure. Enter the minimum and maximum pressure.
Ratio of the Flow Meter to Pipe for Plot and Table	This option allows you to enter specified flow meter to pipe diameter ratios. Enter a ratio and click the Insert button. To remove a ratio, click on it in the list then click the Delete button.
Plot On	Select this to generate a plot for flow rate versus delta pressure for the specified flow meter to pipe diameter ratios.
Table On	Select this to generate a table for flow rate versus delta pressure for the specified flow meter to pipe diameter ratios.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Flow Meter Examples

There are several sample flowsheets in "Chapter 24: Flow Meter Samples - c:\designii\samples\equipmnt\flowmetr" of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.19: Heat Exchanger

General - Single Stream (Air Cooler and Simple)

This Heat Exchanger module can be used to model single-stream heat exchangers. It will also model air-cooled and water-cooled heat exchangers.

General - Two Stream (Complex)

This Heat Exchanger module can be used to model counter-current heat exchange between two streams. You may also specify that rating be done on the Heat Exchanger.

Details

The Heat Exchanger module in DESIGN II can be used to specify heating or cooling of a stream (a single-stream heat exchanger) or to calculate counter-current heat exchange between two streams (a two-stream heat exchanger). The module calculates the heat and material balance from known input stream information and the specification to be met.

Two types of exchangers may be modeled with the single-stream exchanger (see Single-Stream Heat Exchanger for the specification dialogs):

- Single Stream Temperature Change
- Water Cooled Heat ExchangeFor the two-stream exchanger, two additional types of exchangers (Two-Stream Heat Exchanger) can be modeled:

- Counter-Current Heat Exchange
- Refrigerant Heat Exchange

To select the appropriate type, first determine what input information you have, and what information you want to have calculated. For example, do you need all the information about both sides of the exchanger, or are you just trying to set the temperature of your process stream? Do you need to know everything about the heat exchanger now, or could you wait until you finish your design of the process; then go back and get all the information on the exchanger?

The following items should be considered when you specify a model for the heat exchanger:

Stream Order -	Specifications are for the shell-side outlet stream.
Assumed Input Data -	The model assumes that all the data for the inlet stream(s) is known.
Set T and P -	You may set the temperature and pressure of a stream with this module.
Heat Duty Sign -	When specifying a heat duty for the heat exchanger, keep in mind the sign convention.
Dew and Bubble point -	Dew and bubble point feeds require special care to insure that they are the proper phase.

Single Stream Temperature Change
In many process simulations only one side of the exchanger is of interest. For example, you may be interested only in process-stream temperature adjustment, not the amount of the utility required for the heat exchange. For most exchangers that use a heating or cooling substance to adjust the temperature of a process stream, you can model the exchanger with only one stream in and one stream out to reduce the complexity of the simulation.

The duty and/or temperature out of the exchanger are calculated along with all the properties of the outlet stream.

Water Cooled Heat Exchange
Water-cooled exchangers are handled as single-stream exchangers. The water temperatures in and out are specifications for the heat exchanger rather than stream temperature and pressure specifications. The amount of water is calculated as part of the heat exchanger output rather than specified on a flow command or reported in the final stream summaries. Therefore, the water stream in and out of the water cooled exchanger does not need to be numbered nor included as part of the flowsheet. Also, water does not have to be added as a component in the Components dialog in the General specifications.

The water cooled condenser is used in the specific situation where water is used to condense a process stream to the bubble point. This option would normally be used to simulate total condensers associated with columns or where a product must be liquefied for storage.

The amount of water required to condense the process stream is calculated along with the duty, area, corrected LMTD, and outlet stream properties for the process stream.

Counter-Current Heat Exchange
This exchanger simulates two process streams exchanging heat with each other. This usually occurs in a heat recycle situation with one of the streams coming from downstream in the process. For counter-current exchangers, remember to keep your input and output streams in the correct order, make sure your streams are connected so the specification is applied to the correct stream, and review your problem to see if the heat recycle can be removed.

If a simple exchanger is specified, the surface area and overall heat transfer coefficient of the exchanger should also be specified. If not, the area per shell will default to 100 FT2 and the overall heat transfer coefficient will default to 50 BTU/HR/FT2/ °F.

All outlet stream properties are calculated. The duty, area per shell, and corrected log mean temperature difference are reported. This is an iterative process that requires more execution time than a Temperature Out or Duty specification.

Refrigerant Heat Exchange
Refrigerant exchangers are similar to counter-current heat exchangers with the refrigerant entering at the bubble point and leaving at the dew point. Only a single component refrigerant can be handled with the available specifications. Mixed refrigerant exchangers can be handled by entering the refrigerant stream composition as a feed stream using Stream Specification. The Controller module can then adjust the flow through the Stream Manipulator module until a desired heat exchanger or stream property is obtained (such as the process stream temperature out of the exchanger).

Single component refrigerant exchangers require you to add these components in the Components Dialog under the Specify Menu for the refrigerant stream. This allows the flowrate and properties of the refrigerant to be calculated and reported as part of the final stream summary. Also the component identification number of the refrigerant must be included on the Components Dialog.

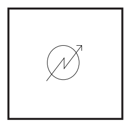
The temperature or pressure of the refrigerant is calculated along with the flowrate required and all properties. The properties of the process stream out are calculated. The duty, area, and corrected LMTD are calculated and reported in the equipment summary.

Please see the online **DESIGN II** Help topic ***Equipment/Heat Exchanger*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 18: Heat Exchanger*** for more details.

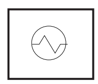
Symbols

The Heat Exchanger module unit has two symbols: Single Stream and Two Stream.

Single Stream: Requires one inlet stream and one outlet stream.



Two Stream: Requires two inlet streams and two outlet streams. The heating or cooling stream can be either a refrigerant stream or other process streams.



Properties (Single Stream Heat Exchanger)

When you double click on the Air Cooler or Single Stream symbol, the equipment properties dialog for this symbol will pop up.

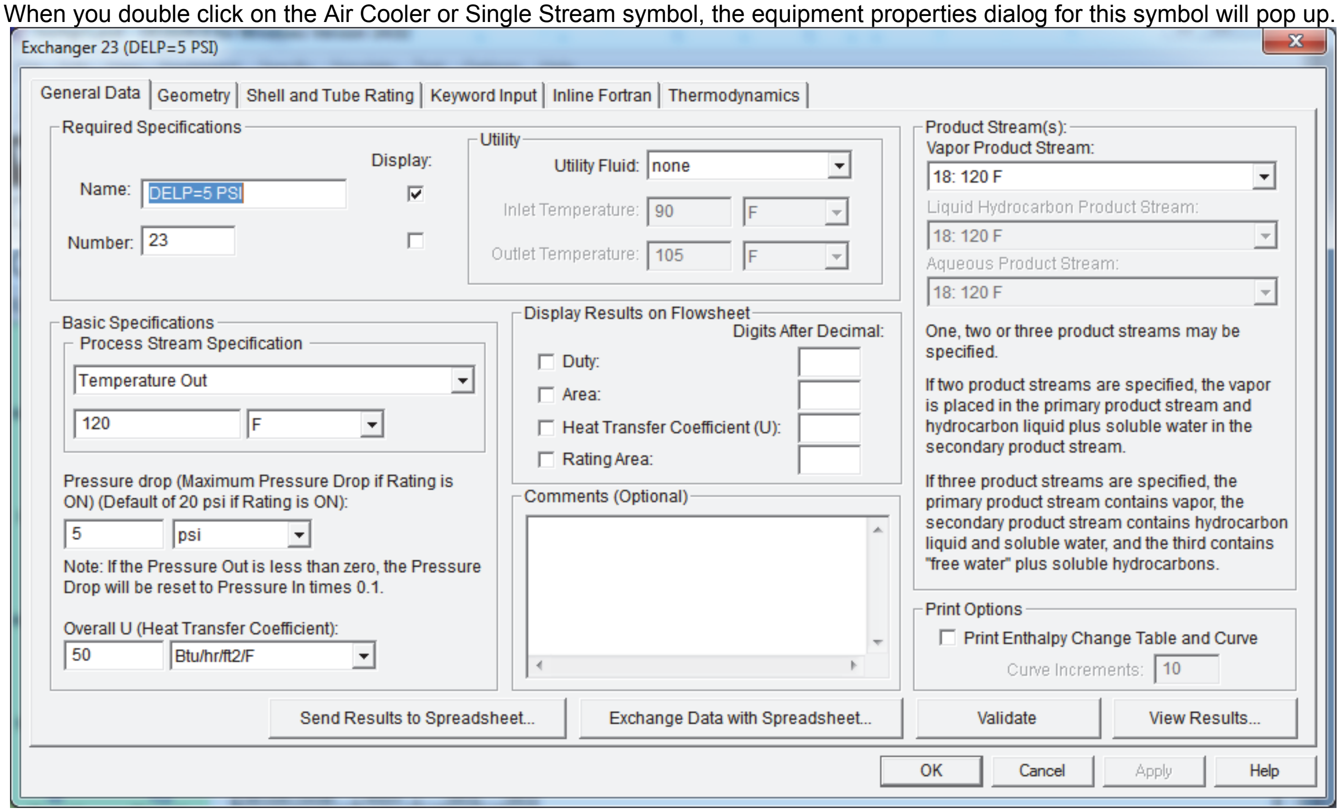


Figure 1: Single Stream Dialog (from exchgr1.psd)

General Data (Single Stream Heat Exchanger)

This dialog provides group boxes, edit boxes, and dialog buttons to enter the basic specifications for the heat exchanger.

Data Item	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Utility	
Utility Fluid	Select a utility fluid, either: None: No specification for the type of utility fluid. DESIGN II will calculate the outlet temperature and/or duty. Water: DESIGN II will calculate the process outlet temperature and water flowrate based on a specified heat transfer coefficient and area. Air: Heat exchanger rating is not allowed with this option.
Inlet Temp	If you selected Water or Air, enter the inlet temperature for the utility fluid and select a unit.
Outlet Temp	If you selected Water or Air, enter the outlet temperature for the utility fluid and select a unit.
Display Results on Flowsheet	You can optionally display the following results on the flowsheet: Duty Area Heat Transfer Coefficient (U) Rating Area If you select to display one or more of these results, enter the number of digits to display to the right of the decimal.
Basic Specifications	
Temp Out	Temperature of the outlet stream; the stream connected to the shell side of the exchanger.
Duty	The heat added to (positive sign) or subtracted from (negative sign) an inlet stream.

UA Exchanger	When you pick this option, you must also select the Geometry button to enter the heat transfer coefficient and/or the area for the exchanger.
Temp Approach	Absolute temperature difference between the shellside outlet stream and the tubeside inlet stream (must be greater than 2 °F).
Delta Temperature	Increase (positive sign) or decrease (negative sign) in temperature of the shellside inlet stream to the temperature of the shellside outlet stream.
Temp Out Bubble Point	Sets the temperature of the shellside outlet stream corresponding to the shellside inlet stream to its bubble point.
Temp Out Dew Point	Sets the temperature of the shellside outlet stream corresponding to the shellside inlet stream to its dew point.
Pressure Drop	Enter the pressure drop.
Overall U	Enter the overall heat transfer coefficient.
Print Options	Click on the Print Enthalpy Change Table and Curve checkbox to include an Enthalpy Change and Curve on the output. You can enter a number of curve increments to use.

Properties (Two Stream Heat Exchanger)

When you double click on the Two Stream symbol, the equipment properties dialog for this symbol will pop up.

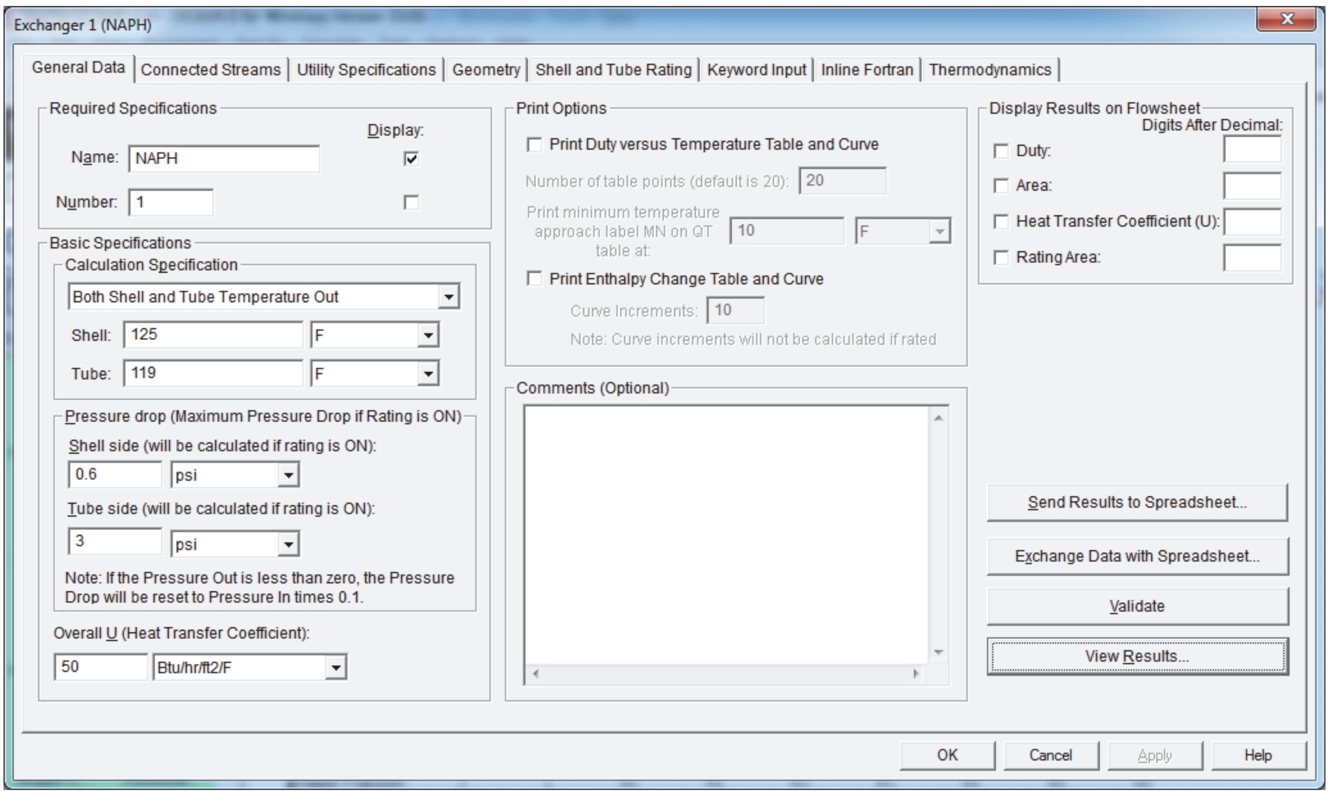


Figure 2: Two Stream Dialog (from exchgr11.psd)

General Data (Two Stream Heat Exchanger)

This dialog provides group boxes, edit boxes, and dialog buttons to enter the basic specifications for the heat exchanger.

Data Item	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Display Results on Flowsheet	You can optionally display the following results on the flowsheet: Duty Area Heat Transfer Coefficient (U)

If you select to display one or more of these results, enter the number of digits to display to the right of the decimal.

Calculation Specification	
Temp Out Shell Side	Temperature of the outlet stream; the stream connected to the shell side of the exchanger.
Duty	The heat added to (positive sign) or subtracted from (negative sign) an inlet stream.
UA Exchanger	When you pick this option, you must also select the Geometry button to enter the heat transfer coefficient and/or the area for the exchanger.
Temp Approach	Absolute temperature difference between the shellside outlet stream and the tubeside inlet stream (must be greater than 2 °F).
Delta Temperature	Increase (positive sign) or decrease (negative sign) in temperature of the shellside inlet stream to the temperature of the shellside outlet stream.
Temp Out Bubble Point	Sets the temperature of the shellside outlet stream corresponding to the shellside inlet stream to its bubble point.
Temp Out Dew Point	Sets the temperature of the shellside outlet stream corresponding to the shellside inlet stream to its dew point.
Temp Out Tube Side	Enter the temperature out of the tube side.

Pressure Drop

Data Item	Description
Shell Side	Enter the shell side pressure drop.
Tube Side	Enter the tube side pressure drop.

Overall U Heat Transfer Coefficient

Data Item	Description
Overall U	Enter the overall heat transfer coefficient.

Print Options

Print Duty versus Temperature Table and Curve	Click this checkbox to include a Duty versus Temperature Table and Curve on the output.
Number of table points	Enter the number of points to use for plotting the curve.
Print minimum temperature approach label MN on QT table at	Enter a temperature and select a unit. This will place the label "MN" on the Q-T table to indicate the minimum temperature approach, based on the value you entered.
Print Enthalpy Change Table and Curve	Click this checkbox to include an Enthalpy Change and Curve on the output.
Curve Increments	Enter the number of curve increments to use.

Connected Streams (Two Stream Heat Exchanger)

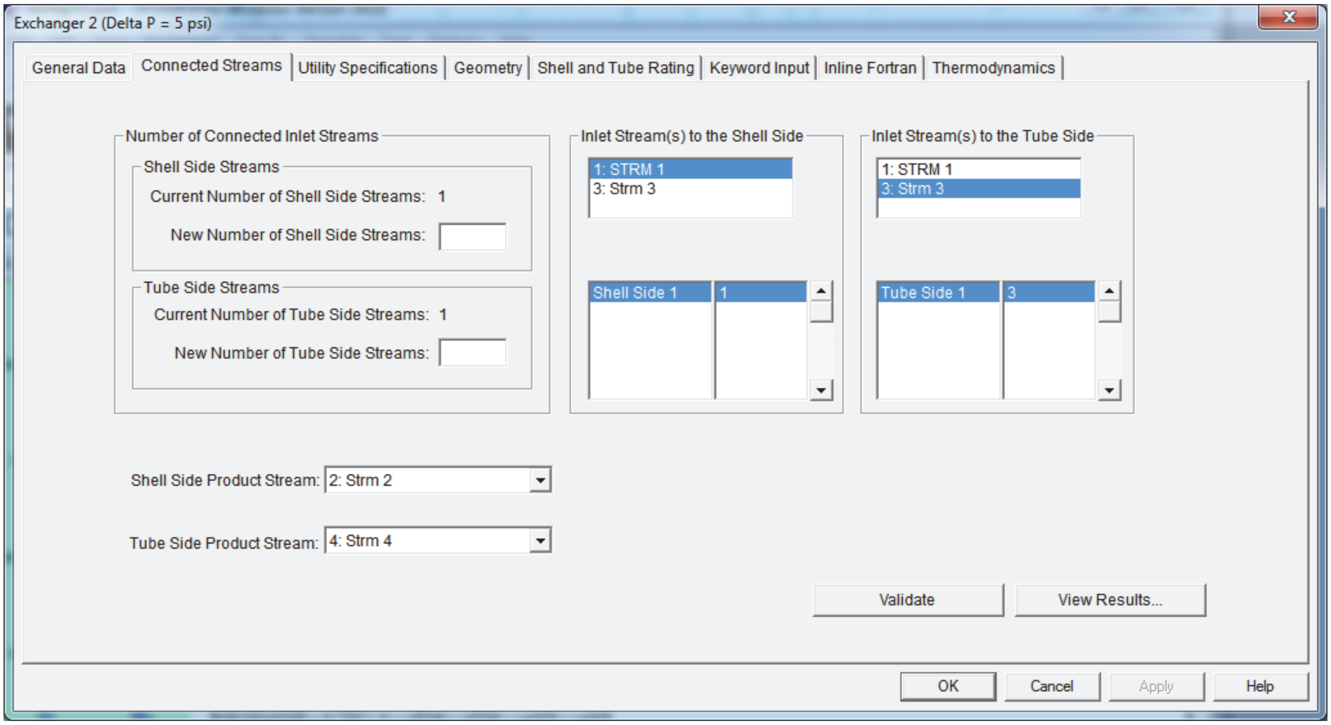


Figure 3: Connected Streams tab (from exchgr11.psd)

Data Item	Description
Number of Connected Streams Shell Side Streams	Enter the new number of shell side streams.
Tube Side Streams	Enter the new number of tube side streams.
Inlet Stream(s) to Shell Side	Click on the shell side from the list then click then click on the stream to which you want to designate at the inlet stream.
Inlet Stream(s) to Tube Side	Click on the tube side from the list then click then click on the stream to which you want to designate at the inlet stream.
Shell Side Product Stream	Open the drop down list and select the stream to use as the shell side product stream.
Tube Side Product Stream	Open the drop down list and select the stream to use as the tube side product stream.

Geometry (Single and Two Stream Heat Exchanger)

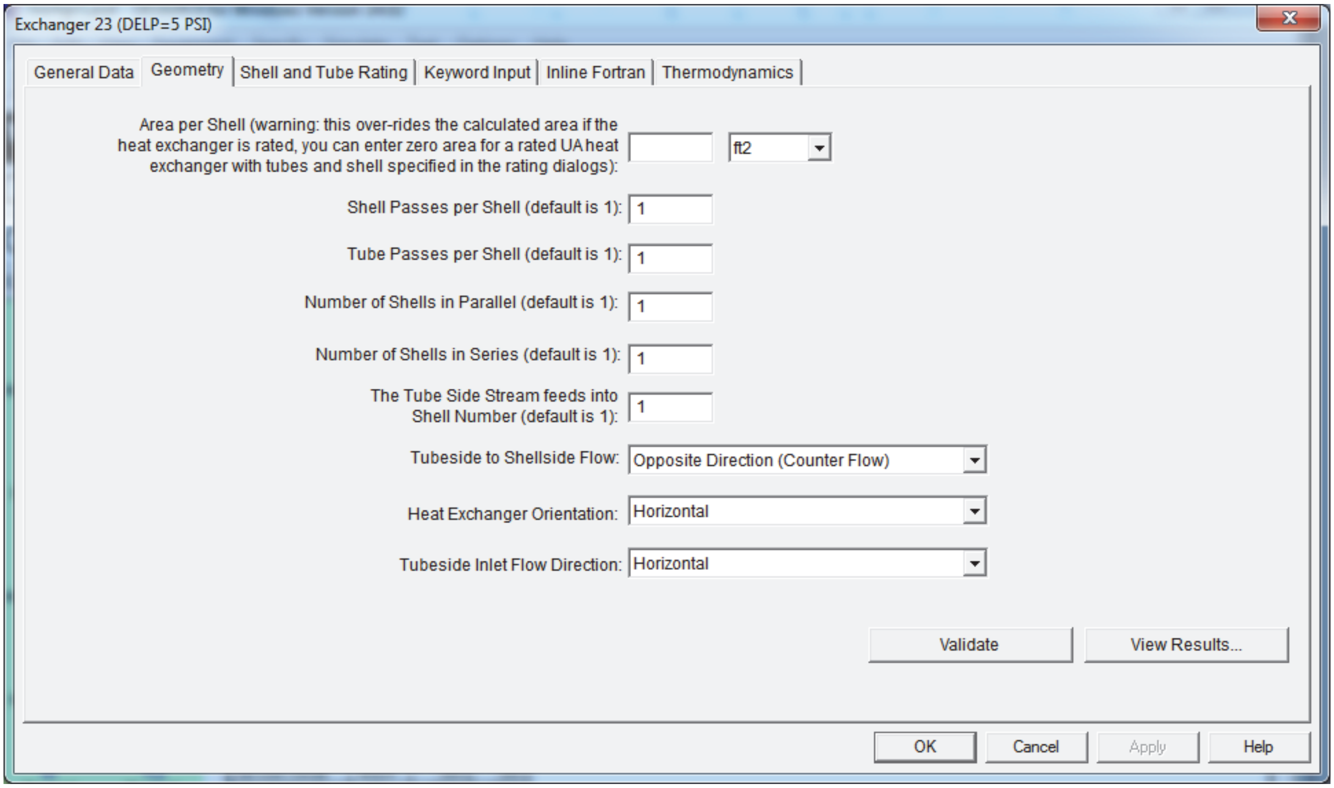


Figure 4: Geometry tab (One Stream and Two Stream) (from exchgr1.psd)

Data Item	Description
Area per Shell	Enter Heat transfer surface area per shell.
Shell Passes per Shell	Enter the number of shell passes per shell.
Tube Passes per Shell	Enter the number of tube passes per shell. Tube Passes is also used by the rating calculation.
Number of Shells in Parallel	Enter the number of parallel shells. For the AES and CHS exchanger types, the number of shell passes is one. For multiple shells, enter a value for the SHElls command. Default is 1.
Number of Shells in Series	Enter the number of series shells.
Tube Side Stream feeds Into Shell Number	Enter the number of the shell to which the tube side stream feeds.
Tubeside to Shellside Flow	Select either Opposite Direction (Counter Flow) or Same Direction (Parallel Flow).
Heat Exchanger Orientation	Select either Horizontal or Vertical (calculations will default to horizontal if shell side is two phase)
Tubeside Inlet Flow Direction	Select: Horizontal Vertical Upwards Vertical Downwards

Utility (Two Stream Heat Exchanger)

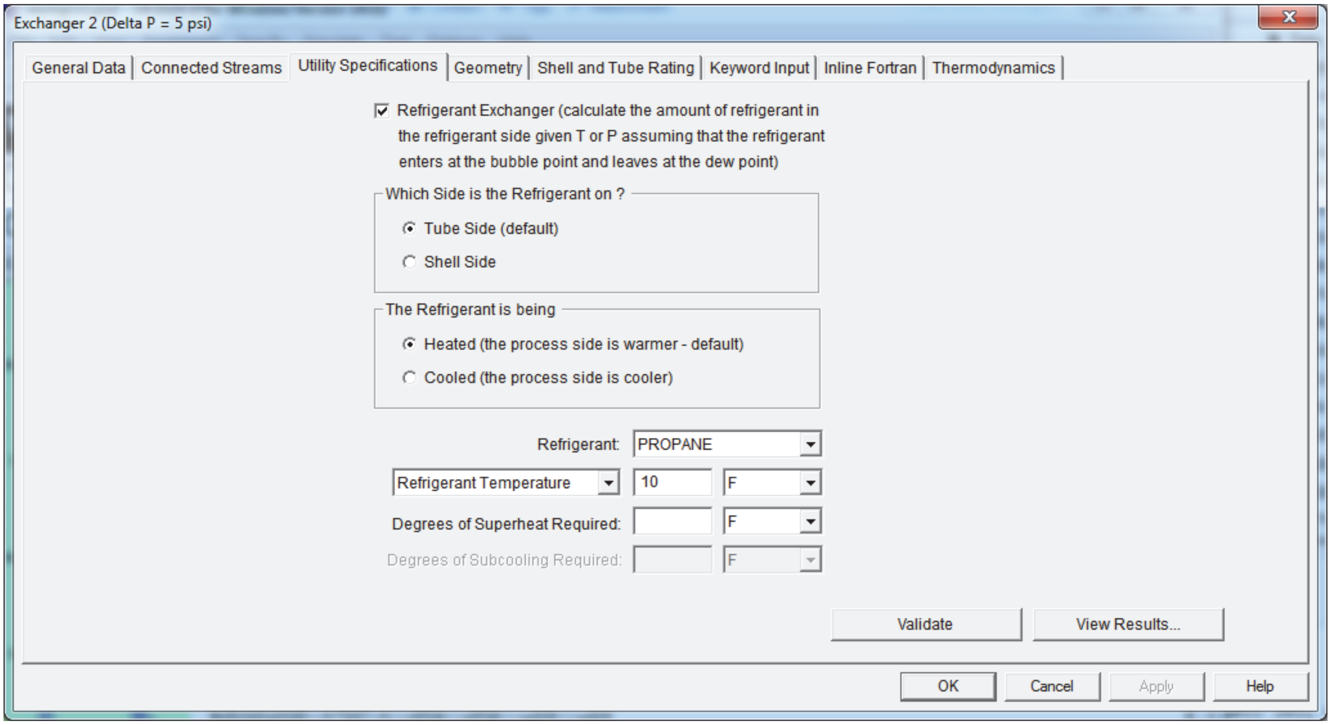


Figure 5: Utility (Single Stream) (from exchgr11.psd)

This dialog allows you to enter refrigerant specifications for a counter-current (two-stream) heat exchanger. This information is required.

Data Item	Description
Refrigerant Exchanger	Select this check box if you are specifying a refrigerant for the exchanger.
Which side is the Refrigerant On	Select either Tube Side or Shell Side
The Refrigerant is Being	Selected either Heated (process side is warmer) or Cooled (process side is cooler)
Refrigerant	This combo box contains a list of the chemical components you have specified (see Components under the Specify menu) for your flowsheet. Select one of these for your refrigerant.
Refrigerant Temperature/ Pressure	Enter the temperature/pressure of the refrigerant. If you select Temperature, the pressure will be calculated; you should enter an initial guess for the pressure as a stream specification for the refrigerant stream (see Setting Stream Specifications in the Streams chapter). If you select pressure, enter the pressure of the refrigerant and select a unit. The temperature will be calculated. You should enter an initial guess for the temperature as a stream specification for the refrigerant stream (see Setting Stream Specifications).
Degrees of Superheat Required	If you selected that the refrigerant is being heated; then enter a temperature and select a unit.
Degrees of Supercooling Required	If you selected that the refrigerant is being cooled; then enter a temperature and select a unit.

Shell and Tube Rating

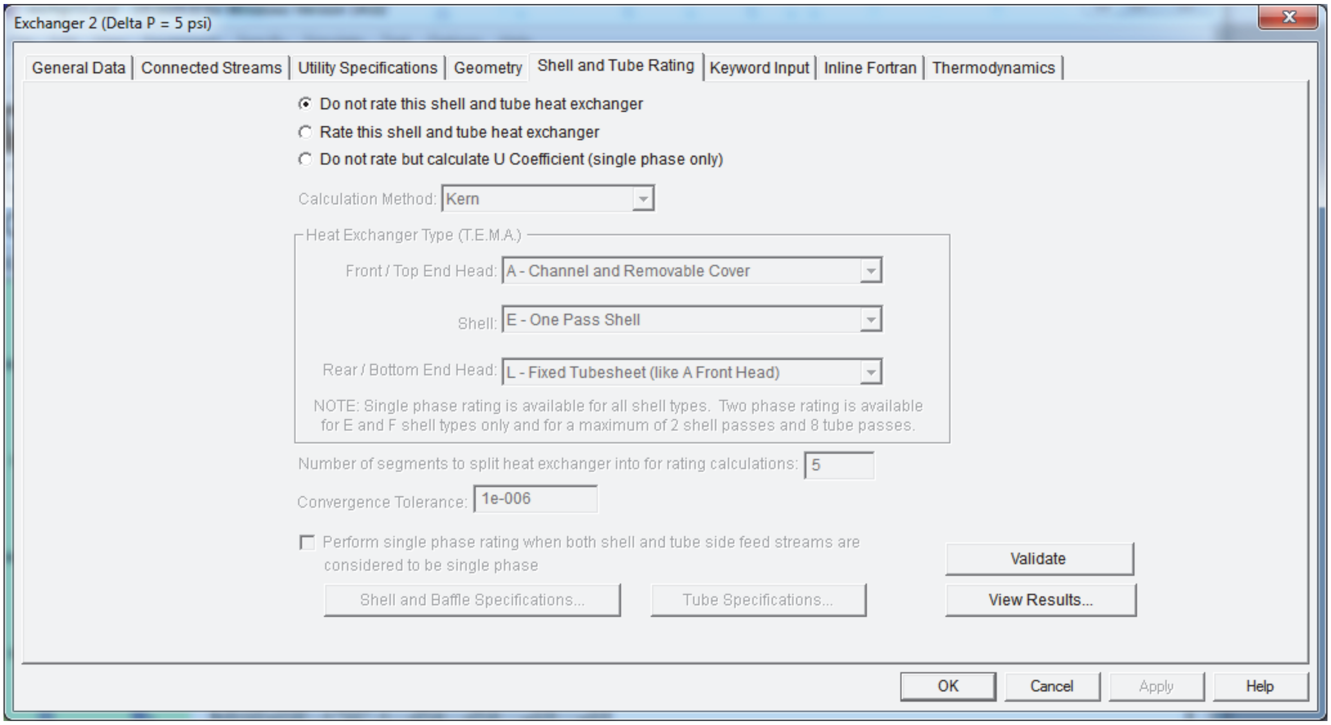


Figure 6: Shell and Tube Rating tab (from exchgr11.psd)

Data Item	Description
Rating	Select either: Do not rate this shell and tube heat exchanger Rate this shell and tube heat exchanger Do not rate but calculate U Coefficient
Calculation Method	Select your choice for the U and pressure drop calculation for both the shell and tube sides.

A single phase rating calculation can be requested for shell-and-tube exchangers for any of the T.E.M.A. types. The two-phase rating calculation accepts only shell types E and F.

Heat Exchanger Type	Specify the exchanger type. Choose one letter for front head, one letter for shell type, and one letter for rear head from standard T.E.M.A. nomenclature as shown in Heat Exchanger section of DESIGN II Reference Guide.
Number of Segments	Specify the number of segments into which the exchanger is to be divided for the rating calculation. The accuracy of the rating is increased by increasing the number of increments.
Convergence Tolerance	Enter the heat exchanger rating convergence tolerance to be achieved.
Shell and Baffle Specifications	Click this button to specify shell and baffle data.
Tube Specifications	Click this button to specify tube data.

Shell and Baffle Specification

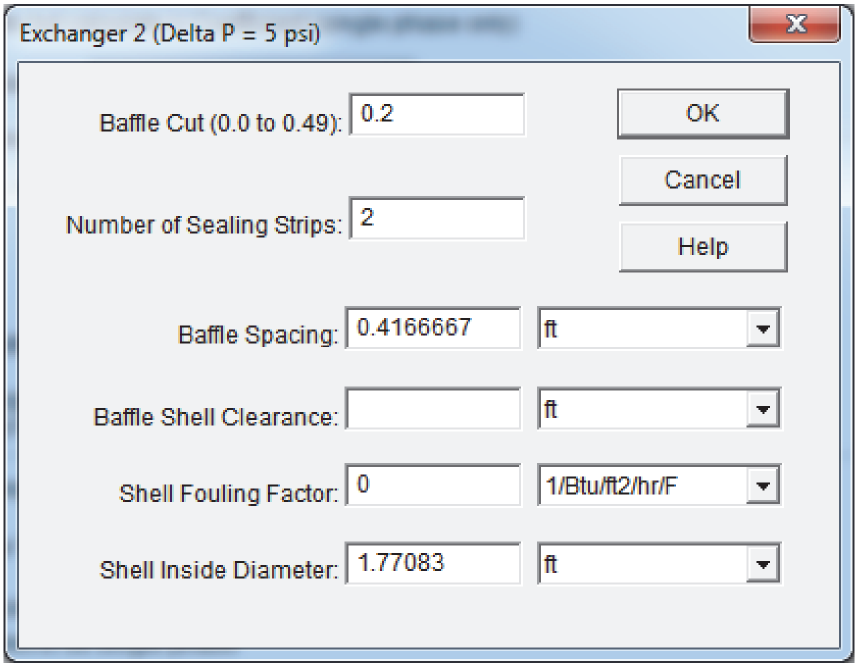


Figure 7: Shell and Baffle Specifications (from exchgr11.psd)

Data Item	Description
Baffle Cut	Enter the fractional value for the baffle cut. Default is 0.2.
Number of Sealing Strips	Sealing strips are added to baffles to prevent bypass leakage. This value is only for inclusion in the printout and is not used in the rating calculation.
Baffle Spacing	Enter the baffle spacing.
Baffle shell clearance	Enter the clearance between the shell and baffles. This command is only used with the Bell method. If no value is specified, it will be calculated from the geometry.
Shell fouling factor	Enter the shell fouling factor. A fouling factor for the shell may be entered which will affect the value of the overall heat transfer coefficient.
Shell inside diameter	Enter the inside diameter of the shell.

Tube Specification

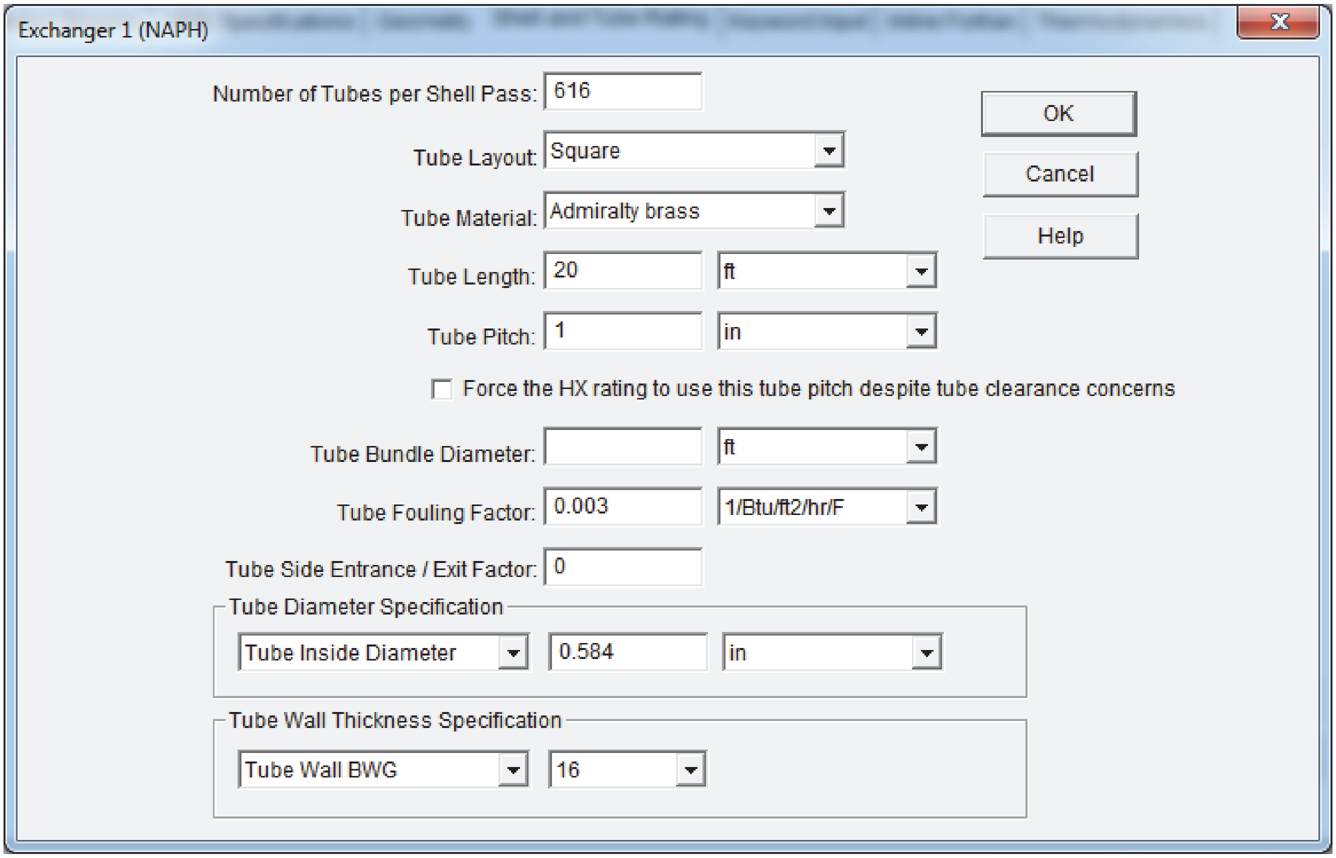


Figure 8: Tube Specifications (from exchgr12.psd)

With the addition of a RATing command and appropriate commands for shell-and-tube geometry, DESIGN II will perform a rating calculation on any single phase counter-current exchanger (all TEMA types) and certain two-phase exchangers (E and F series only) in your flowsheet. The commands are in the same, simple English-language format as other HEAt EXChanger commands. Most of the geometry commands have reasonable default values if you choose not to enter your own.

The rating calculations are performed for shell-and-tube exchangers. DESIGN II first performs the regular HEAt EXChanger calculation using the specifications you have entered to calculate outlet stream temperatures and total duty. All required properties for the rating calculation, such as viscosities, specific heats, thermal conductivities, densities, phase composition, and mass flows are automatically calculated for inlet and outlet conditions of both streams.

For 2-phase heat exchangers, the RATing option of the HEAt EXChange module simulates the phase change along the longitudinal direction of the exchanger by dividing the exchanger into several sections.

In each section DESIGN II performs flashes and determines the two-phase flow regime. The module then chooses a heat transfer correlation that is best suited to the conditions and performs heat transfer and pressure drop calculations. Currently, the two-phase rating calculation accepts only two shell types: E and F, and the following exchanger configurations: 1-1, 1-2, 1-4, 2-2, 2-4, 2-6, 2-8. However, single phase rating is available for all shell types. The only multiple shell heat exchanger configuration that can be rated in series is type E and only two shells may be specified.

Data Item	Description
Number of Tubes	Enter the number of tubes per shell pass.
Tube Layout	Enter your choice for tube layout.
Tube Material	Enter choice of tube material.
Tube Length	Enter the length of the tubes. Standard tube lengths are 8, 10, 12, 16, and 20 feet with 16 feet as the most common length.
Tube Pitch	Enter the tube pitch. Tube Pitch is defined as the shortest center-to-center distance between two adjacent tubes. The shortest distance between two tubes is called clearance.
Tube Bundle diameter	Enter the diameter of the tube bundle. This command is only used for the Bell method. If no value is entered, it will be calculated from the geometry.

Tube Fouling factor	Enter the fouling factor for the tubes. The tube fouling factor, if specified, will be used in the calculation of the overall heat transfer coefficient.
Tube inside diameter	Enter the inside diameter of the tubes. If no value is entered, it will be calculated.
Tube outside diameter	Enter the outside diameter of the tubes.
Tube BWG	Enter the Birmingham Wire Gauge for tube wall thickness.
Tube wall thickness	Enter the wall thickness of the tubes. If no value is entered, the program will calculate it.

If more than the above four tube dimensions are entered, the program will use the pair that is highest in the previous list. If only one of the above commands is entered, the following command will be used to complete the pair.

IF	THEN
Tube Wall thickness	Tube outside diameter of 0.75 will be used
Tube inside diameter	Tube wall thickness of 0.065 will be used
Tube outside diameter	Tube wall thickness of 0.065 will be used
Tube BWG	Tube outside diameter of 0.75 will be used

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Thermodynamics (Single Stream)

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II Reference Guide.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Heat Exchanger Examples

There are several sample flowsheets in “Chapter 25: Heat Exchanger Samples - c:\designii\samples\equipmnt\heaexc” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.20: Hydrotreater

General

The Hydrotreater module is a yield model of a unit for hydrogenation of olefinic and aromatic crude feeds and for desulfurization and denitrogenation of high-sulfur crude. The reactor is an adiabatic type with catalyst beds packed in series. Quench streams are added between beds to absorb the large amount of heat released by the exothermic reactions. The quench streams can be either the hydrogen feed or reactor products (minus hydrogen, hydrogen sulfide, and ammonia). The outlet stream from the hydrotreater is a combination of the product stream, the quench stream(s) and excess hydrogen. A second exit stream can be coded if you want to see the quench flowrate and composition.

Your COMponents command should include hydrogen, hydrogen sulfide and ammonia in addition to the components which make up your feed. You can enter as many hydrocarbon feed streams as you wish. The last stream entered must be the hydrogen stream. The next-to-last stream should be the hydrocarbon product flowrates on a component-by-component basis, excluding quench, excess hydrogen, ammonia, and hydrogen sulfide.

Details

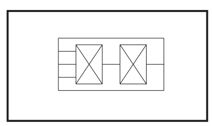
If more than one hydrocarbon feed is present (number of input streams is greater than 3), the hydrocarbon feeds are mixed. The hydrogen stream (composition and flowrates have been specified by the user) is then mixed with the hydrocarbon feed(s). If the resulting hydrogen partial pressure is too low for the reactor, the program calculates a new flowrate for the hydrogen stream; composition remains the same, total flowrate changes.

The program uses the extent of reaction and feed information (specified by the user) to determine the heat released by the reactions as well as flowrates for ammonia and hydrogen sulfide. The program makes iterative calculations around each catalyst bed, determining the required quench flowrate, the percent completion of the reactions, and the heat of reaction for each bed. The outlet stream from the hydrotreater will consist of the hydrocarbon product plus quench and excess hydrogen.

Please see the online **DESIGN II** Help topic ***Hydrotreater*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 19: Hydrotreater*** for more details.

Symbols

The Hydrotreater unit module has one symbol:



The Hydrotreator module requires that at least three inlet streams and at least one outlet stream be connected to the module.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

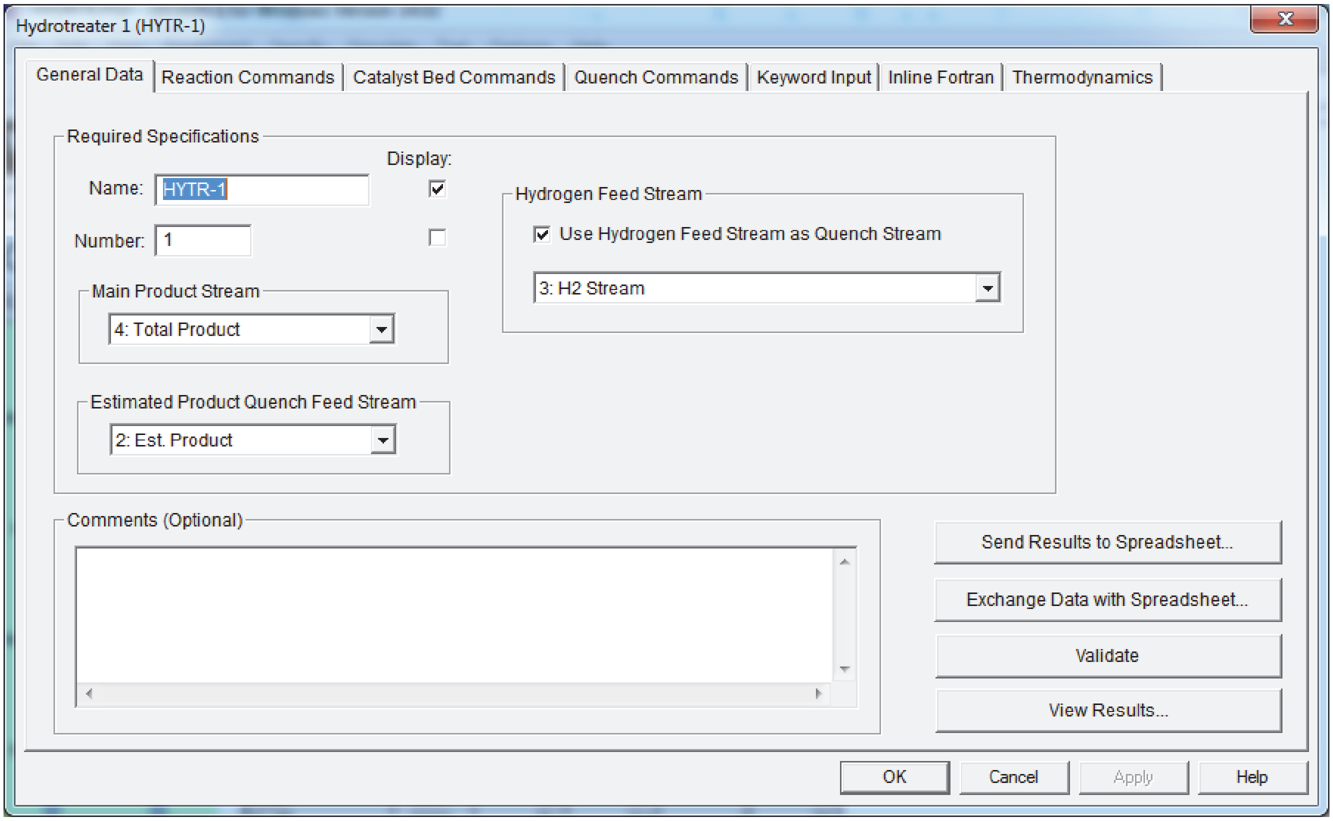


Figure 1: Hydrotreator Dialog (from hydrtr.psd)

General Data

Data Item	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Main Product Stream	Select the desired stream to use as the main product stream.
Estimated Product	Select the desired stream to use as the estimated product quench feed stream.
Use Hydrogen Feed As Quench Stream	Hydrogen Feed Stream as Quench Stream: Select this checkbox to specify a hydrogen quench. Then, select the desired stream to use as the hydrogen feed stream.

Reaction Commands

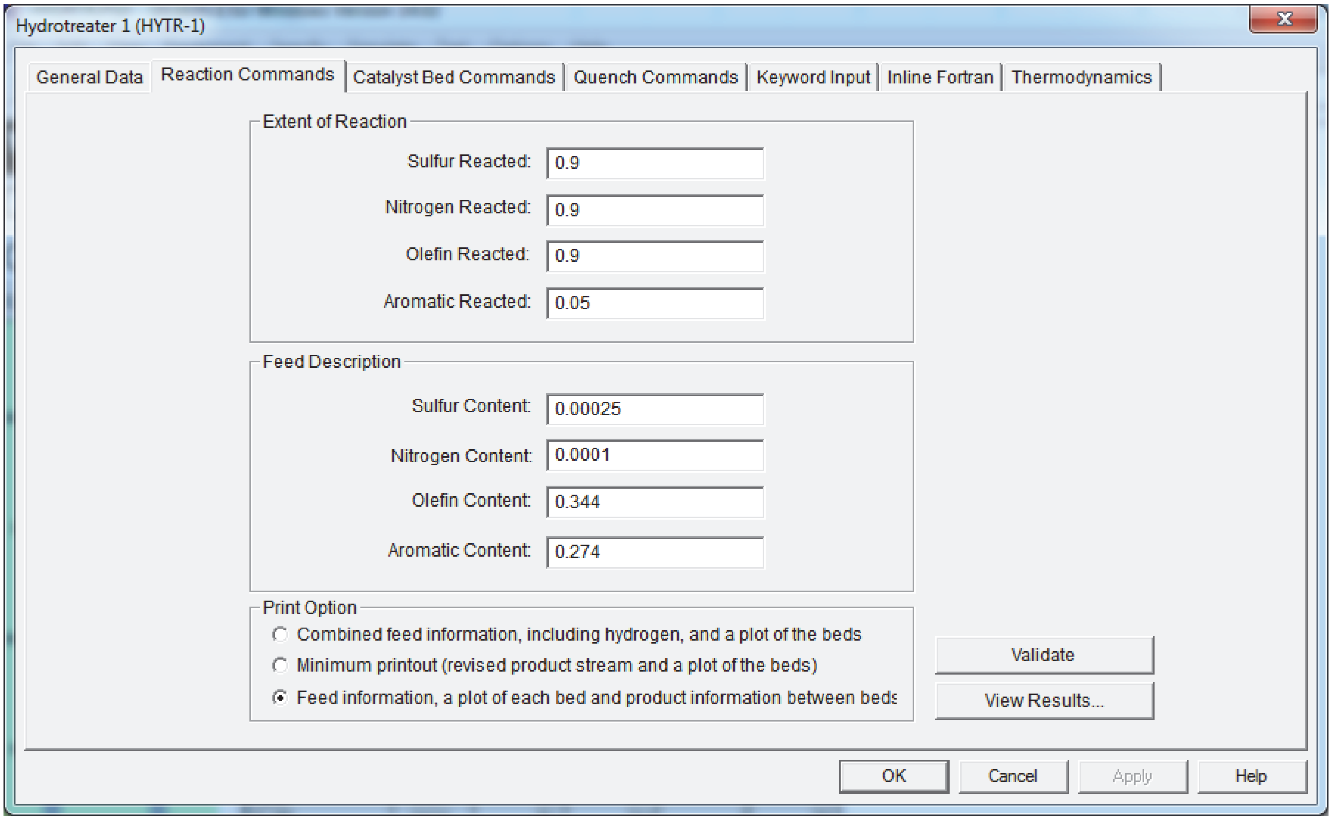


Figure 2: Reaction Commands tab from Hydrotreater Dialog (from hydrtr.psd)

Data Item	Description
Extent of Reaction	
Sulfur Reacted	Enter the weight fraction of sulfur converted to hydrogen sulfide
Nitrogen Reacted	Enter the weight fraction of nitrogen converted to ammonia
Olefin Reacted	Enter the mole fraction of olefins converted to saturated hydrocarbons
Aromatic Reacted	Enter the mole fraction converted to saturated hydrocarbons
Feed Description	
Sulfur Content	Enter the weight fraction of sulfur in the feed stream
Nitrogen Content	Enter the weight fraction of nitrogen in feed stream
Olefin Content	Enter the mole fraction of olefins in feed steam
Aromatic Content	Enter the mole fraction of aromatics in feed stream.
Print Option	Select one of the following: Combined feed information, including hydrogen, and a plot of the beds. Minimum printout (revised product stream and a plot of the beds). Feed information, a plot of each bed and product information between beds.

Catalyst Bed Commands

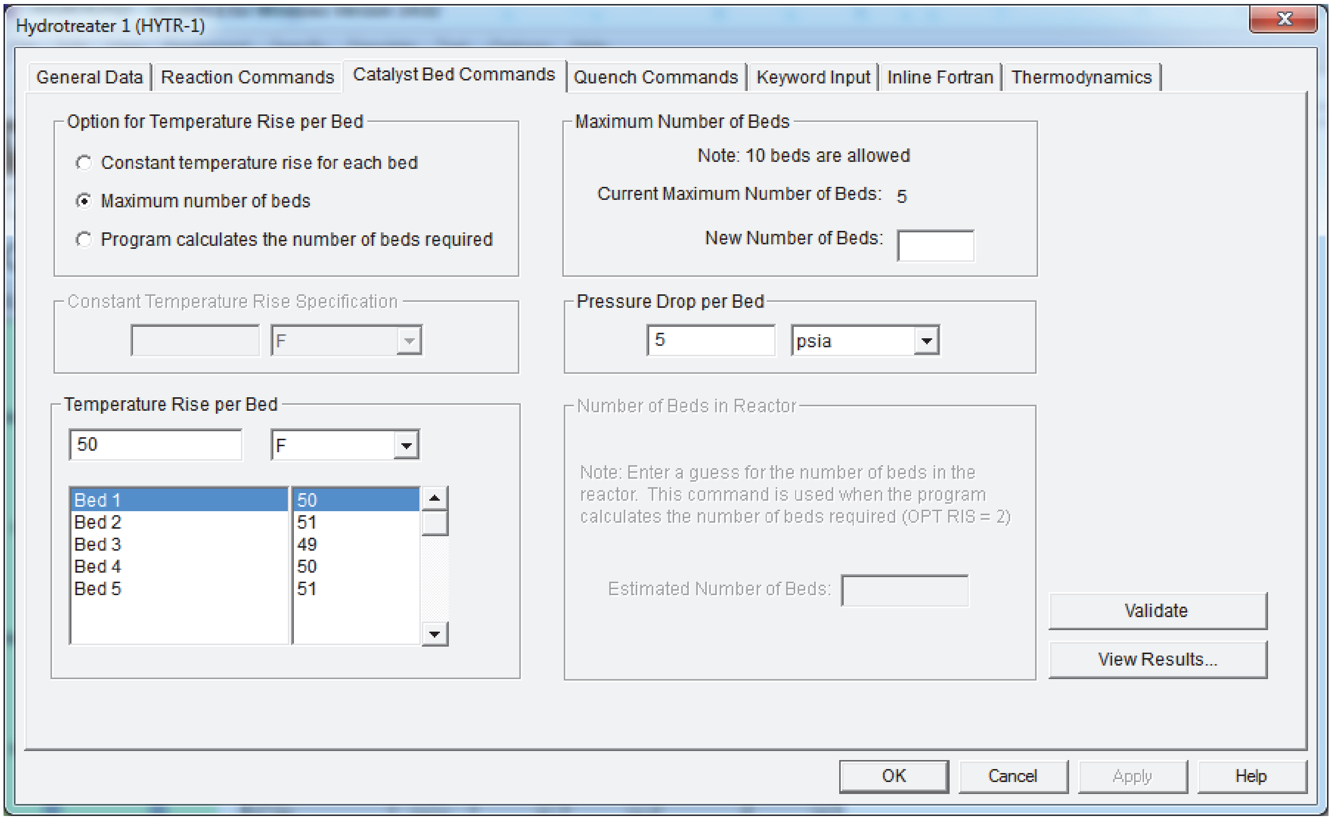


Figure 3: Catalyst Bed Commands tab from Hydrotreater Dialog (from hydrtr.psd)

Data Item	Description
Option Temperature Rise per Bed	Select either: Constant temperature rise for each bed Maximum number of beds Program calculates the number of beds required
Maximum Number of beds	Current Maximum Number of Beds: View the number.
New Number of Beds	Enter the new number of beds to use. 10 beds are allowed.
Constant Tempeature Rise Specification	Enter the permissible temperature rise between beds.
Pressure Drop per Bed	Enter the pressure drop per bed.
Temperature Rise per Bed	Enter the temperature rise per bed.
Number of Beds in Reactor	Enter a guess for the number of beds in the reactor.

Quench Commands

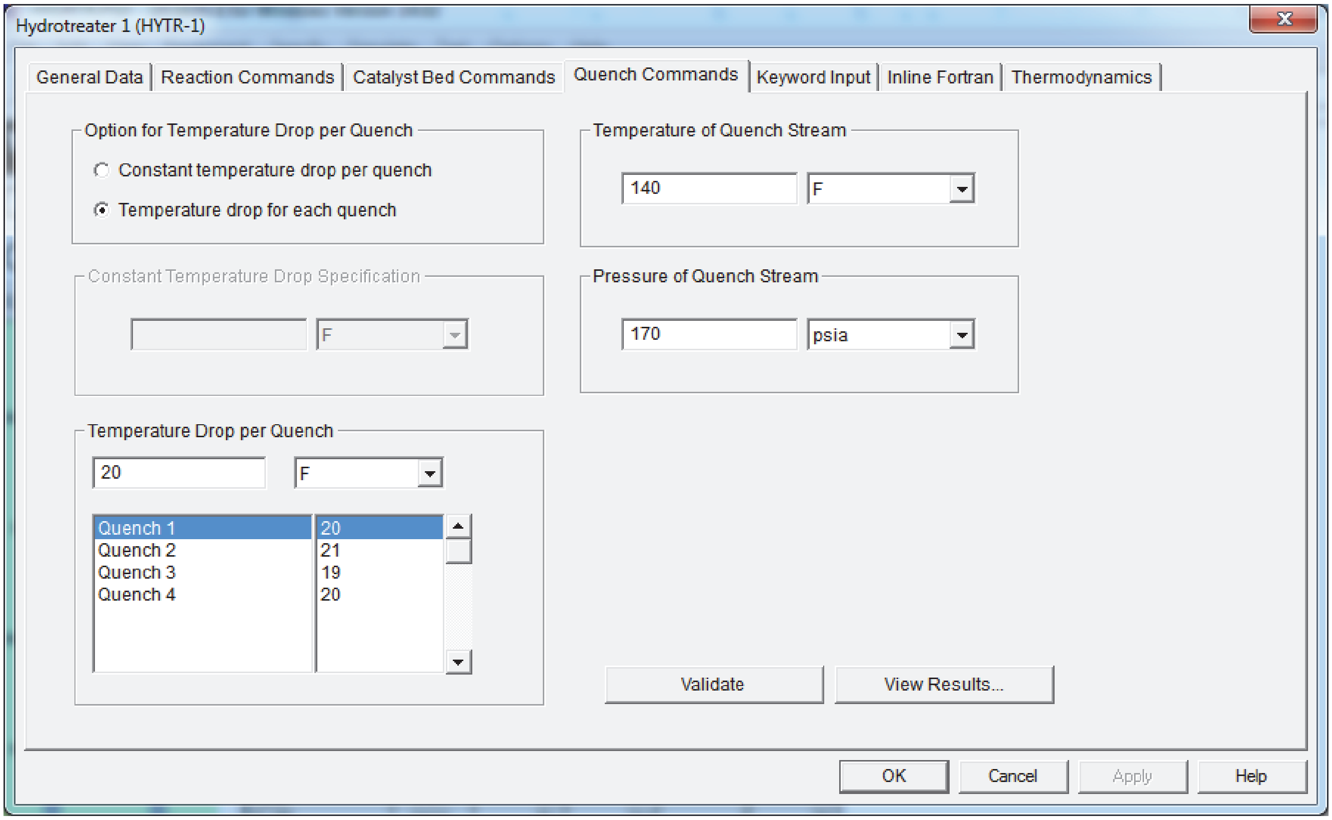


Figure 4: Quench Commands tab from Hydrotreater Dialog (from hydrtr.psd)

Data Item	Description
Options for Temperature Drop per Quench	Select either: Constant temperature drop per quench Temperature drop for each quench
Temperature of Quench Stream	Enter the temperature of the quench stream.
Constant Temperature Drop Specification	If you selected Constant temperature drop per quench, enter the temperature drop between beds to use.
Pressure of Quench Stream	Enter the pressure of the quench stream.
Temperature Drop per Quench	Select each quench; then enter the temperature drop per quench.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Hydrotreater Example: Crude Treating

There are several sample flowsheets in “Chapter 26: Hydrotreater Samples - c:\designii\samples\equipmnt\hytr” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.21: Line Pressure Drop

General

Use the Line module to calculate pressure drops due to friction and elevation changes (if any) in transmission lines or plant piping of specified length and diameter.

Details

The Line module calculates pressure drops due to friction and elevation changes (if any) in transmission lines or plant piping of specified length and diameter. The Line module must have one inlet stream and one outlet stream. There are two snap points for the Line module and either may be used for the inlet or the outlet. The inlet stream to the Line module may be gas, liquid, or two-phase.

Calculations can be isothermal, adiabatic, or based on heat transfer to the surroundings. The heat transfer to surroundings may be specified by user-supplied heat transfer coefficients or may be estimated by supplying the properties of the surrounding medium. You can select several calculation methods. The method options currently available are primarily applicable to hydrocarbon and hydrocarbon-water systems.

The following topics provide background on setting up the Line module:

- Line Description
- Methods of Calculation
- Thermodynamic Guidelines

Line Description

The Line Pressure Drop module works by dividing the Line into several smaller segments of different lengths and performing a calculation on each segment. Sections of the Line are determined by breaking the Line at points where property changes occur. The properties of a Line may either be interval or point.

Interval properties are those that have a constant value over a specified interval but a different value in different intervals.

The following are Interval properties:

Elevation Change	Temperature of Surroundings
Overall Heat Transfer Coefficient	Conductivity of Surroundings
Velocity of Surroundings	Pipe Depth
Insulation Thickness	Insulation Conductivity

Point properties are those that have a specific value at a specific point on the line. The value of the property on points between the specified points is either zero or interpolated depending on the context.

Point properties are Absolute Elevation (interpolated) and Equivalent Length of fittings (a zero value between the points specified).

Since the segments are divided based on the points where property changes occur, it is possible that some segments can be very long; it can increase convergence difficulties for flash calculations. More importantly, the accuracy of the pressure drop calculation decreases for very large pressure drop values. You can easily prevent this by specifying the maximum segment length (or fraction of total length).

Methods of Calculation

The calculation methods used are primarily applicable to hydrocarbon systems in fairly small diameter lines. Results will vary with the density and viscosity correlations you choose for the simulation. Also, results of two-phase and non-isothermal calculations will vary with the K-value and enthalpy correlations, respectively.

The correlations for two phase flow are largely empirical, and their accuracy varies considerably, even within the range of the correlated data. Use of these methods for situations other than those intended by the authors is not recommended.

You are allowed to select calculation methods for modeling both two-phase and single phase behavior in the pipeline. The program will determine the phase of the mixture entering the pipeline and use one of your selections.

If the phase changes during calculations, DESIGN II will switch to the other set of methods. The primary method should be a two-phase method; the alternate selection should be for vapor. If you choose not to select any methods for calculating pressure drop in the pipeline, DESIGN II will use the default options shown in the tables below.

An X under horizontal or vertical indicates that orientation is not allowed.

Methods for Friction Loss - Primary & Alternate

Two-Phase Orientation

Method	Horiz	Vert	Description
Beggs and Brill			Integrated method for pressure drop, holdup, and flow regime for large diameter, long pipelines with horizontal or inclined orientation
Dukler			Uses the constant slip method
Duns-Ros	X		Integrated method for pressure drop, holdup, and flow regime for vertical pipe
Hagedorn-Brown	X		Recommended for vertical liquid systems
Lockhart-Martinelli	X		Default for 2 phase horizontal
Mukherjee-Brill			Integrated method for pressure drop, holdup, and flow regime for all pipe orientations
Orkiszewski	X		Default for 2 phase vertical
Oliemans	X		Correlation for gas condensate systems

Single Phase Orientation

Method	Horiz	Vert	Description
American Gas Association			For turbulent gas flow
Darcy-Weisbach*			Default for single phase horizontal; accuracy decreases for P > 10% of inlet pressure
Panhandle A			For long, gas, large diameter lines
Modified Panhandle			Modified for GPSA
Weymouth			Conservative for short gas lines

*Darcy-Weisbach is the default method for single phase systems- gas or liquid.

Methods for Friction Factor - Primary & Alternate

Two-phase Orientation

Method	Horiz	Vert	Description
Jain			For flow regime with Reynolds numbers of 5E3 to 1E8
Moody			Default for two-phase systems, corresponding to the Moody chart

Single Phase Orientation

Method	Horiz	Vert	Description
American Gas Association			Default with MET FRI=AGA, for turbulent gas flow
Jain			For flow regime with Reynolds numbers of 5E3 to 1E8
Moody			Default with MET FRI=DARcy Weisbach, corresponds to the Moody chart
Panhandle A			Default with MET FRI= PAN or MOD PAN; predicts lower than Moody or Weymouth
Weymouth			Used with MET FRI= WEY; based on air in small diameter pipes

Methods for Holdup Two-phase Orientation

Method	Horiz	Vert	Description
Beggs Brill			
Dukler			
Duns-Ros	X		
Eaton			
Hughmark & Pressburg			
Lockhart-Martinelli			
Mukherjee-Brill			
Hughmark			

Methods for Elevation Two-phase Orientation

Method	Horiz	Vert	Description
American Gas Association	X		To be used with AGA method of friction
Flanigan			For long transmission lines in hilly terrain.
Orkiszewski*	X		For vertical upwards flow in the slug flow regime
Phase Density			Default, recommended for most simulations

- Vertical up only.

Thermodynamic Guidelines

DESIGN II inlet stream information provides physical properties of fluid; thus, you do not need to enter them. Subsequent isothermal flashes (default), adiabatic flashes, or heat exchange calculations are performed at each of the Line nodes and phase properties are calculated.

The thermodynamic options you choose from the Basic Thermo Dialog are used to generate K-values and enthalpies. For heat and material balance calculations, the K-values and enthalpies are the most important properties. For calculating pressure drops in lines, other physical properties- particularly density and viscosity- are also important.

Some guidelines for density choices follow:

- The Corresponding States Density method is used for natural gas mixtures up to about 400 F, if the mixture does not contain large amounts of CO2 or H2S;
- The Benedict-Webb-Rubin equation of state is used for most hydrocarbon and inorganic gases at high pressures;
- The Yen-Woods (STD) correlation is used for most other systems, including mixtures such asmethanol and water.

NOTE: Both Corresponding States and Benedict-Webb-Rubin methods are based on density data and can be slow in execution.

The default viscosity option is NBS 81 (Ely-Hanley, National Bureau of Standards; however, if Assay Data is specified the default option is API. NBS 81 is recommended for predicting low molecular weight hydrocarbon vapor and liquid viscosities; it is not suitable for polar mixtures.

The Dean and Stiehl technique was developed for estimating vapor viscosities; it does not estimate liquid viscosities well and should not be used for LINE calculations.

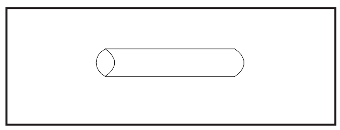
The API technique is the default viscosity option when your input specification includes Assay Data, Refine columns or petroleum streams. If the feed contains significant amounts of light components (methane through pentanes, CO2, H2S, and N2), you may want to use the NBS 81 option instead. Simply select NBS 81 as the viscosity option from the Basic Thermo Dialog.

Sonic velocity in the gas phase is estimated using the generalized correlation of T.K Sherwood. Each Line node is checked to make sure that the flow is subsonic and sonic velocity is reported at the outlet conditions.

Please see the online **DESIGN II** Help topic ***Equipment/Line Pressure Drop*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 20: Line Pressure Drop*** for more details.

Symbols

The Line Pressure Drop unit module has one symbol:



The Line module requires that you connect one inlet and one outlet stream to the module.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

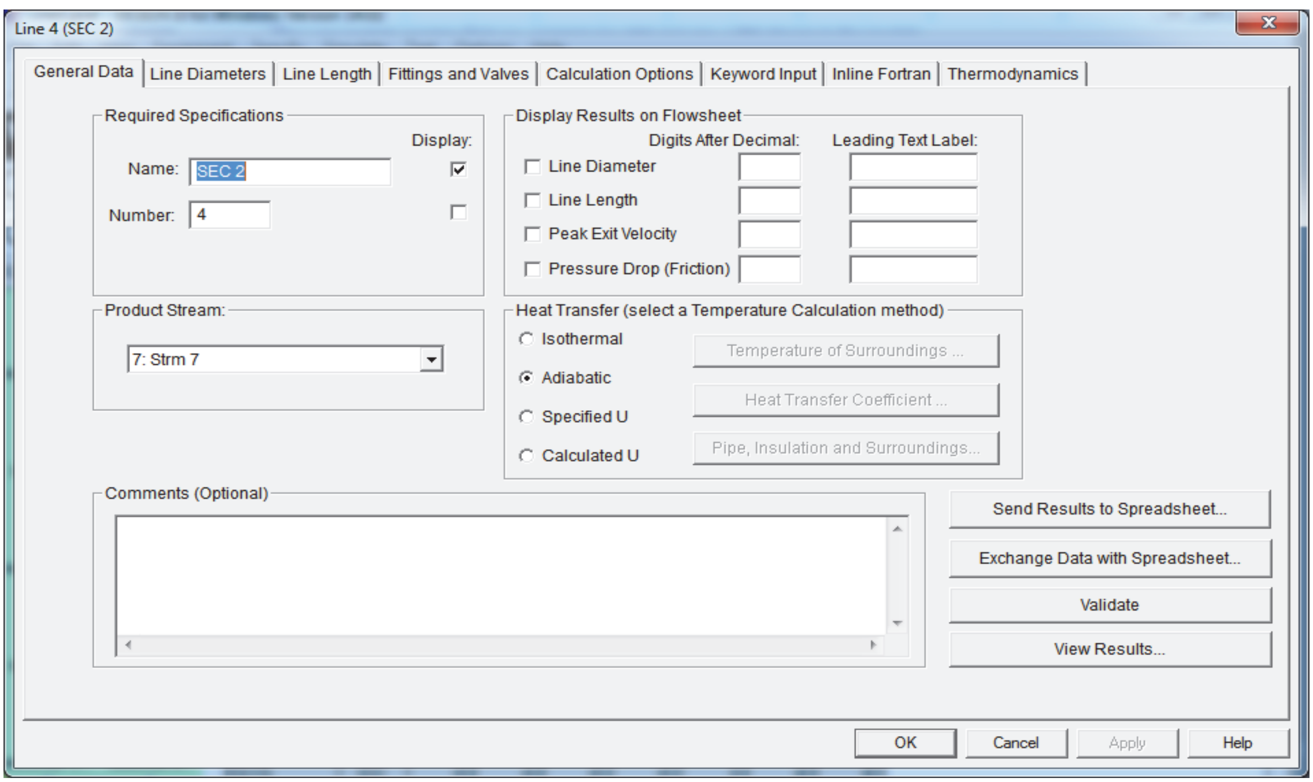


Figure 1: Line Dialog (from line1.psd)

General Data

Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Product Stream	Select the stream from the drop down list to use as the product stream.
Display Results on Flowsheet	You can optionally display the following results on the flowsheet: Line Diameter Line Length Peak Exit Velocity If you select to display one or more of these results, enter the number of digits to display to the right of the decimal, along with any text you might want to display before the results.

There are four radio buttons for selection of the Temperature Calculation Method. The options are:

- Isothermal — No temperature change (the default).
- Adiabatic — No heat loss.
- Specified U — Provide the temperatures of the surroundings and heat transfer coefficients using cascading dialog boxes (click the Temperature of Surroundings and Heat Transfer Coefficients buttons to view the related dialogs).
- Calculated U — Provide the temperatures of the surroundings and details about the pipe, insulation and surroundings, using cascading dialog boxes. DESIGN II uses this information to calculate the heat transfer coefficients (click the Temperature of Surroundings and Pipe, Insulation, and Surrounds buttons to view the related dialogs).

Line Diameters

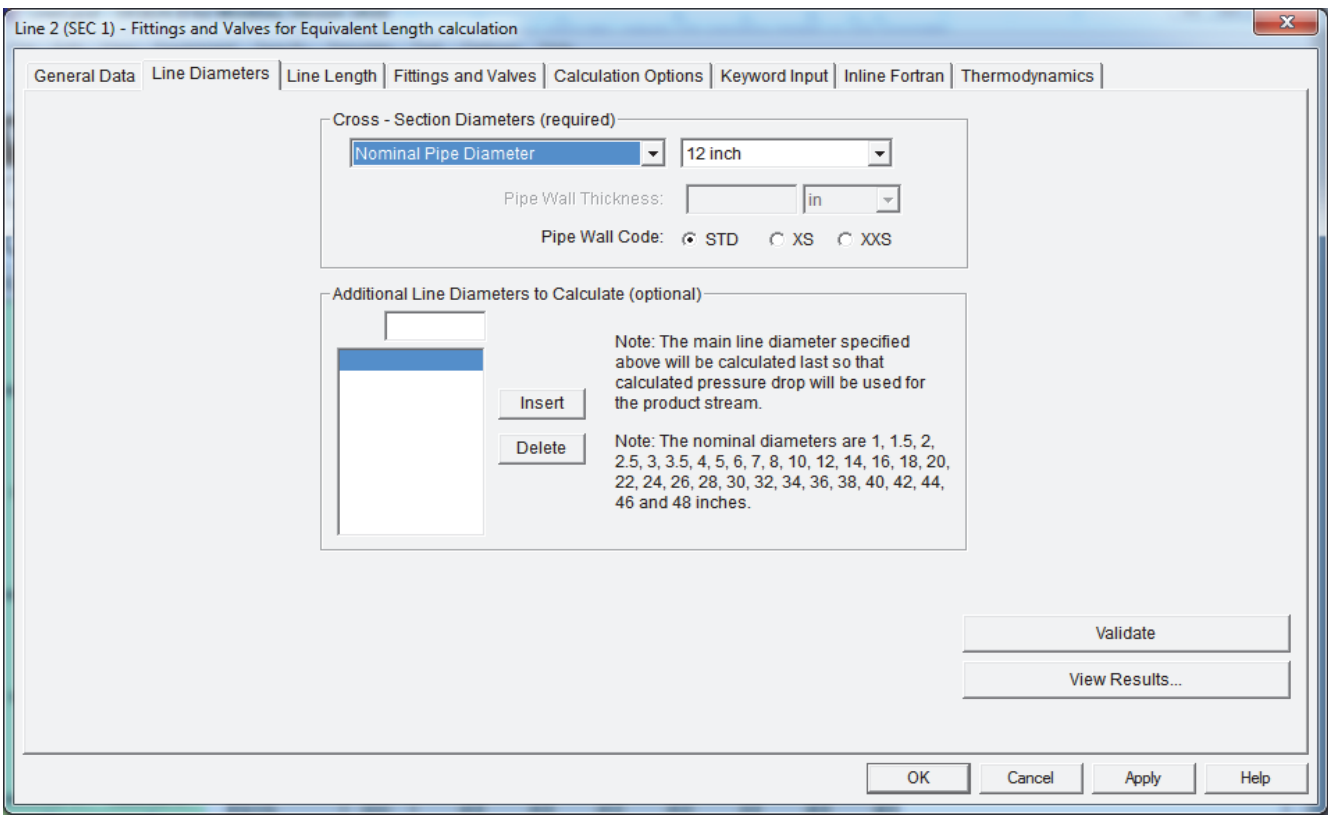


Figure 2: Line Diameters tab from Line Dialog (from line1.psd)

Cross Section

Use this group box to specify the inside diameter and pipe wall thickness of the Line. These specifications may either be made by a Nominal Pipe Diameter and a Pipe Wall Code or by Inside Pipe Diameter and Pipe Wall Thickness.

The contents of the group box will change depending on the way you specify the diameter in the combo box. If you select Nominal Pipe Diameter from the combo box you must select one of the nominal diameters from the adjacent combo box and a Pipe Wall Code from one of the three radio buttons. If you select Inside Pipe Diameter you must enter an inside diameter in the adjacent edit box and enter a Pipe Wall Thickness in the edit box below.

Data Item	Description
Nominal Pipe Diameter	Nominal Pipe Diameter based on ANSI B36.10 and B36.19
Inside Pipe Diameter	Actual inside diameter of the pipe
Pipe Wall Code	Codes which indicate the thickness of the pipe wall: STD = Standard XS = Extra Strong XXS = Double Extra Strong Available only if Nominal Pipe Diameter is specified.
Pipe Wall Thickness	Actual thickness of the pipe wall. Available only if Inside Pipe Diameter is specified.

Line Length

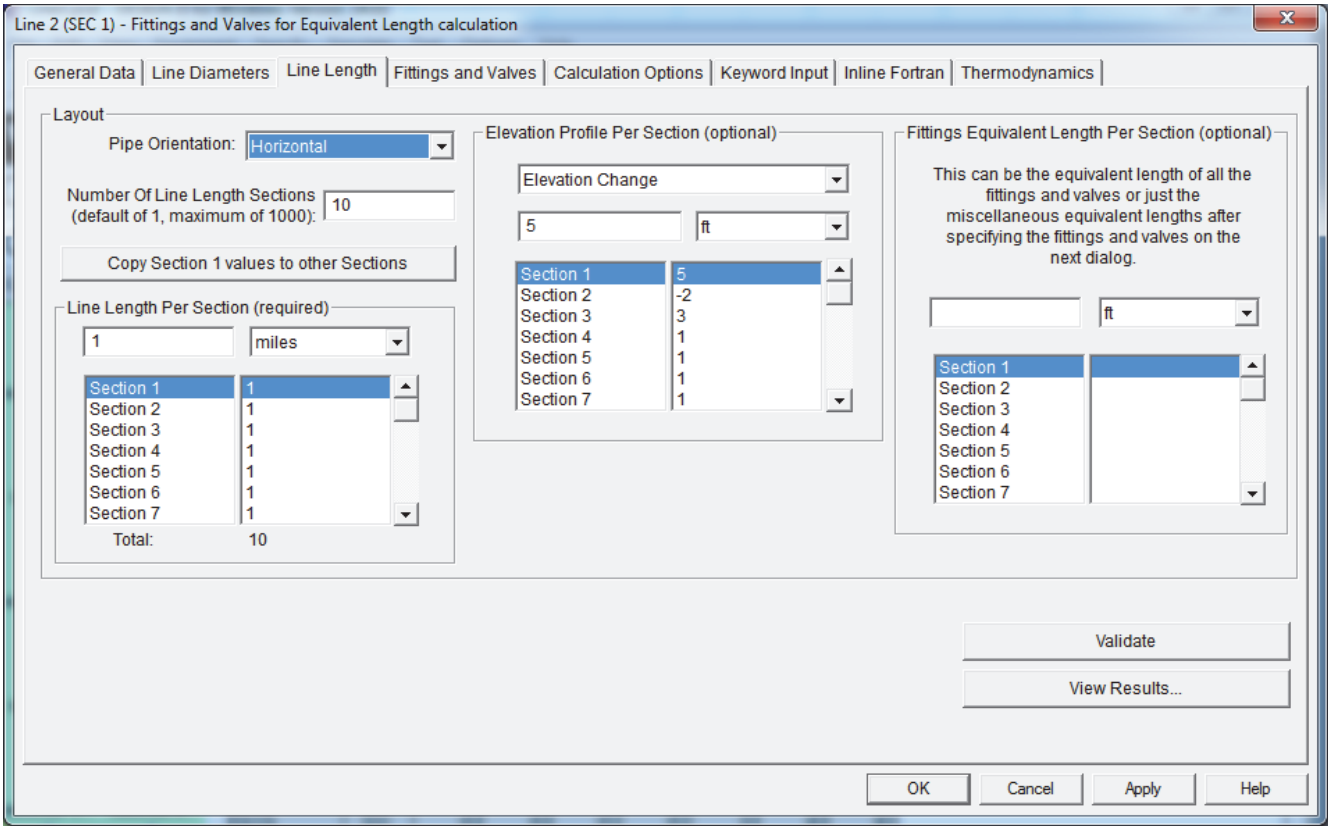


Figure 3: Line Length tab from Line Dialog (from line1.psd)

Layout

This group box is used to specify the orientation and length of the Line.

The Pipe Orientation is specified using a combo box. The choices are Horizontal or Vertical Upwards or Downwards.

Enter the Total Line Length in the edit box then select dimensional units from the adjacent combo box. Total Line Length is the actual length of the Line exclusive of fittings.

Elevation changes and additional equivalent length due to fittings are entered via list boxes according to the number of pipe segments. These specifications are optional. If no elevation change is entered the Line is not inclined from horizontal or vertical.

Data Item	Description
Orientation	The Line is horizontal or inclined The flow is vertical or inclined in an upward direction The flow is vertical or inclined in a downward direction
Total Line Segments	Number of Line segments for the Pipe
Elevation Profile	This listbox allows you to specify elevation changes along the length of the line. Specify the elevation by selecting Elevation or Elevation Change from the combo box; select No Elevation Change if there is no change in elevation.
Elevation (absolute)	Pipe elevations at elevation measurement positions along the Line. The initial elevation can be set by the user.

Elevation Change	Relative changes in elevation for specified elevation change intervals. Positive values indicate an increase in elevation, while negative values indicate a decrease in elevation.
Fittings Equivalent Length	Use this dialog to specify additional equivalent length due to fittings along the length of the Line.

Fittings and Valves

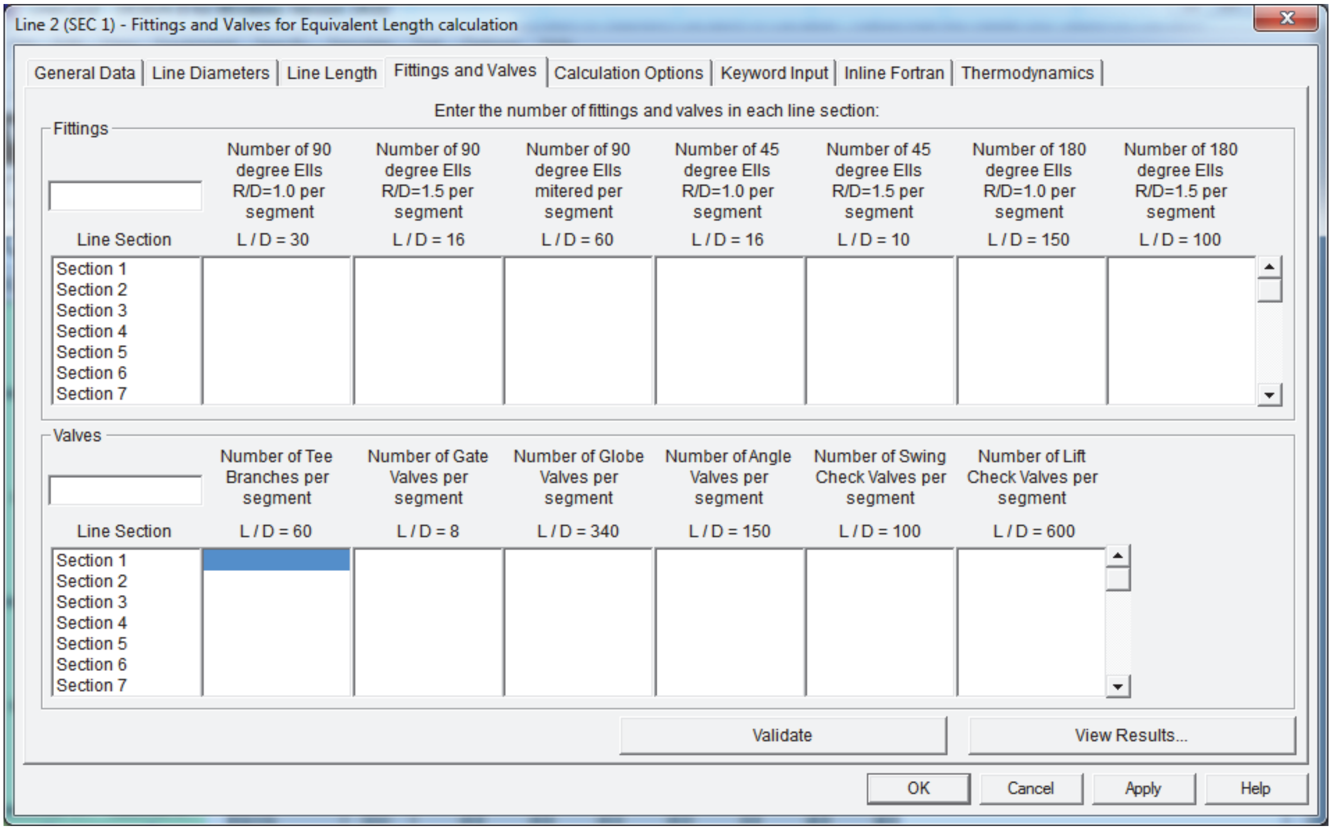


Figure 4: Fittings and Valves tab from Line Dialog (from line1.psd)

Fittings
This group box is used to specify the number of fittings in each line section.

Valves
This group box is used to specify the number of valves in each line section.

Calculation Options

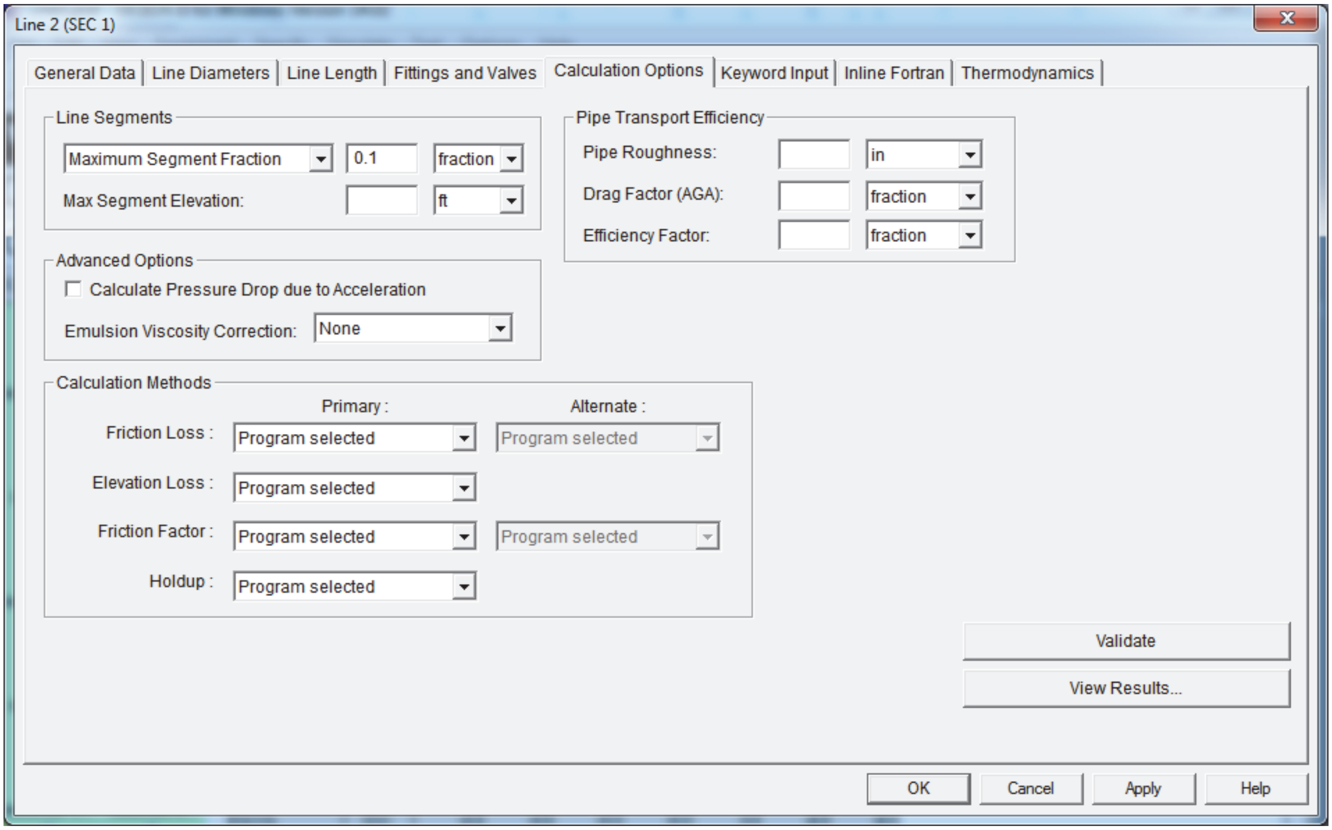


Figure 5: Calculation Options tab from Line Dialog (from line1.psd)

Line Segment
This group box is for specifying the maximum length of a segment and the maximum elevation of a segment to the simulator. The maximum length of a segment may be specified as:

No Max Segment Limit	Use property changes to divide the line into segments;
Max Segment Length	The maximum length of a segment is specified in length units;
Max Segment Fraction	The maximum length of a segment is defined as a fraction or percent of the total length of the line.

The maximum elevation of a segment must be entered in length units.

Data Item	Description
Max Segment Length	Specifies maximum length of a segment as an absolute length, a fraction or percent of total length, or no maximum segment length.
Max Segment Elevation	Specifies maximum elevation of a segment as an absolute elevation

Pipe Transport Efficiency
Use this group box to specify the:

- Pipe Roughness- in length units,
- Drag Factor [AGA]- as a fraction, and
- Efficiency Factor- as a fraction.

These commands are used by the calculation methods for gas (single phase) lines.

Data Item	Description
Pipe Roughness	Specifies absolute or effective roughness in length units.
Drag Factor	Used to determine the transition Reynolds number and to predict the AGA friction factor when flow is partially turbulent.
Efficiency Factor	Specifies the efficiency of the Line as a fraction.

Advanced Options:
This group box is used to:

- Calculate Pressure Drop due to Acceleration (click a check box)

- Emulsion Viscosity Correction (use the adjacent combo box); choosing None indicates no emulsion is formed in the Line between hydrocarbons and water.

Data Item	Description
Pressure drop due to Acceleration	Requests acceleration effect calculation
Emulsion Viscosity	Specifies correction as Loose, Medium, or Tight.

Calculation Methods
You can select a primary and an alternate method for calculating Friction Loss and Friction Factor. This allows you to specify one method for two phase calculations and another method for single phase calculations. DESIGN II evaluates the phase at each node of the Line module and switches between these methods as needed.

If you choose "Program Selected", the method used depends on the phase, flow direction, and flow regime.

The Friction Factor Method usually defaults to the Friction Loss pressure drop method if the "Program Selected" option is chosen. Several of the gas phase methods have a corresponding friction factor method. Use caution when mixing frictional pressure drop options and friction factor methods.

Data Item	Description
Friction Loss Method	Specify Primary & Alternate method
Elevation Loss Method	Specify Primary method
Friction Factor Method	Specify Primary & Alternate method
Holdup Method	Specify Primary method

Temperature of Surroundings

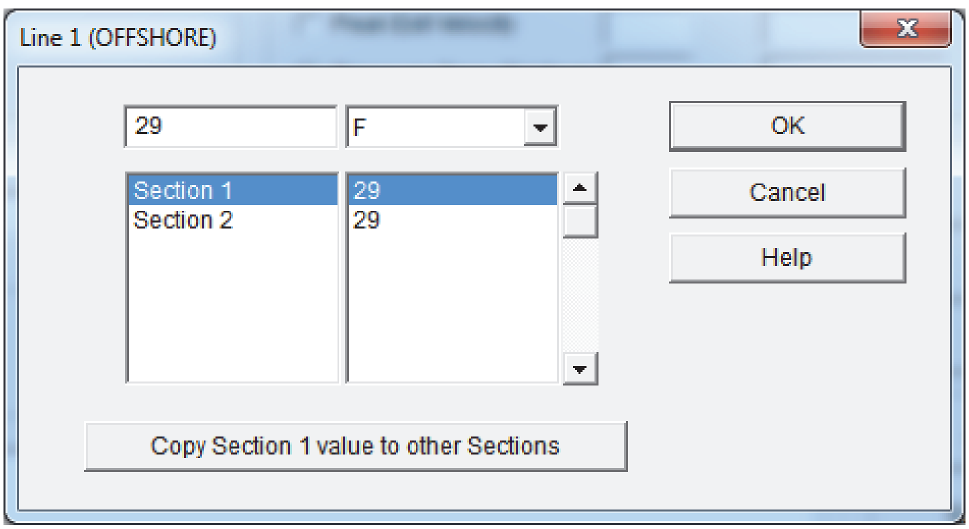


Figure 6: Surroundings Temperature (from line4.psd)

Temperature of Surroundings
Use this dialog to specify the temperature of the environment around the Line at each line segment.

Data Item	Description
Surroundings Temperature	Temperature of surroundings for the specified intervals

Heat Transfer Coefficient

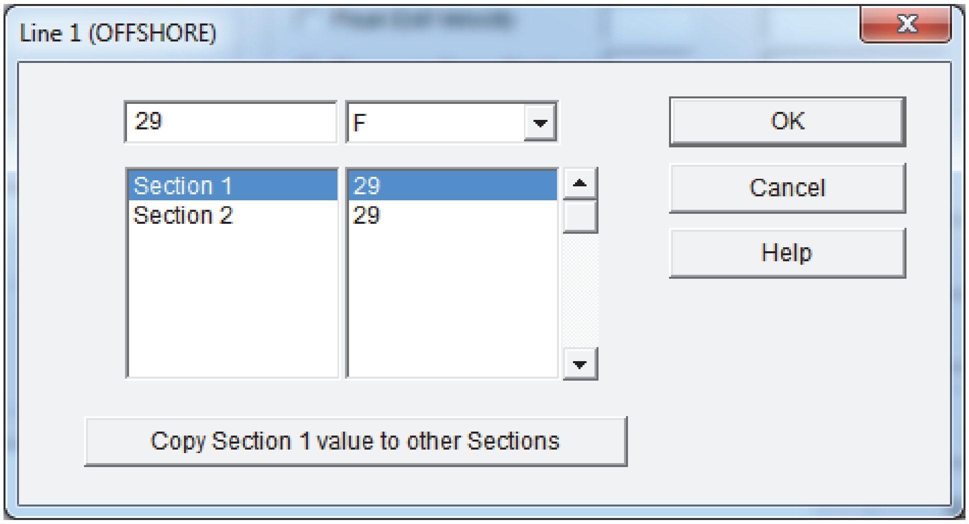


Figure 7: Heat Transfer Coefficient (from line4.psd)

Heat Transfer Coefficient
This dialog is used to specify the overall heat transfer coefficient for each segment of the Line.

Data Item	Description
Heat Transfer Coefficient	Overall heat transfer coefficient for the specified intervals.

Pipe, Insulation, & Surroundings

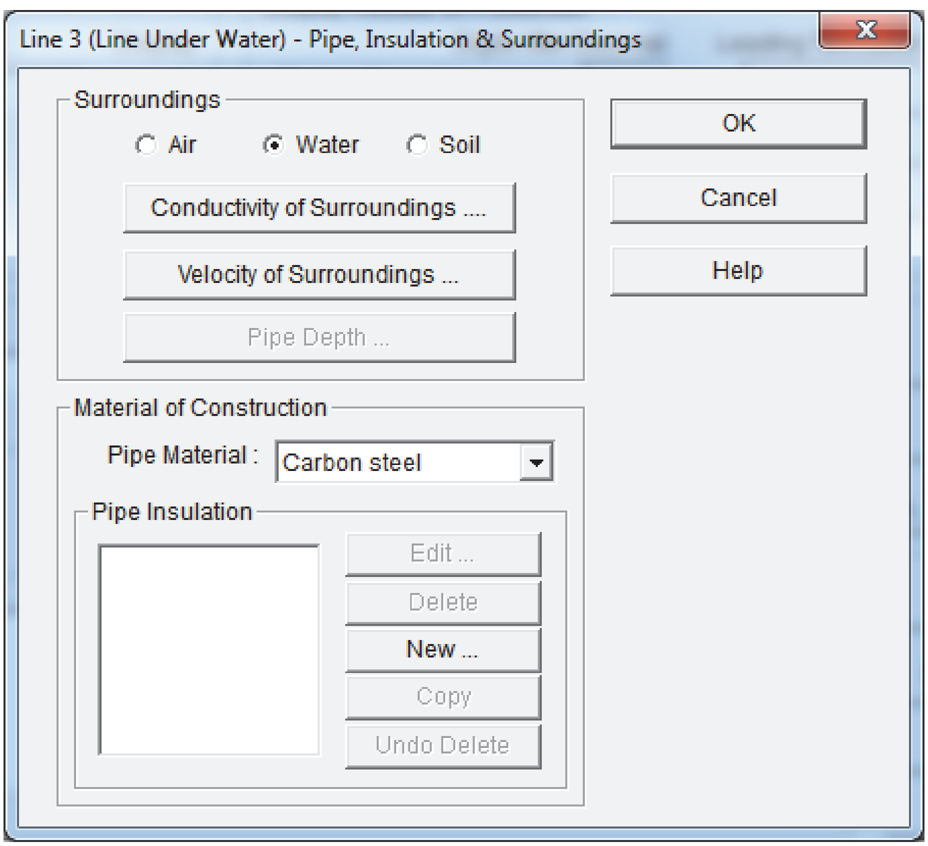


Figure 8: Pipe, Insulation & Surroundings (from line6.psd)

Pipe, Insulation and Surroundings
Use this dialog to specify the surroundings and the materials of construction of the Line.

Surroundings
This group box has three radio buttons- for selecting Air, Water, or Soil as the surrounding environment for the entire Line.

When you choose Air, Water, or Soil, two cascade dialog boxes in the group box become active. The specific boxes are listed under each choice:

Air-	Conductivity of Surroundings / Velocity of Surroundings
Water-	Conductivity of Surrounding Water / Velocity of Surroundings
Soil-	Conductivity of Surrounding Soil / Pipe Depth

For Conductivity of Surroundings Air and Conductivity of Surroundings Water, you can accept the defaults or enter your own values.

The default for Conductivity of Surroundings Air is 0.015 btu/hr.ft.F, while the default for Conductivity of Surroundings Water is 0.3517 btu/hr.ft.F

Material of Construction
This group box is used to specify the material of construction of the Line. The Pipe Material may be specified as any one of ten options from the adjacent combo box. Carbon steel is the default. The pipe insulation is specified through a cascade dialog box.

Data Item	Description
Pipe Material	One of the following may be selected: Admiralty brass, Alloy 825, Aluminum, Carbon steel, Carbon Moly Steel, Copper, Copper nickel 70/30, Copper nickel 90/10, Monel, Nickel, Red brass, Stainless steel 304, Stainless steel 214, Titanium
Pipe Insulation	Click to specify pipe insulation.

Conductivity of Surroundings

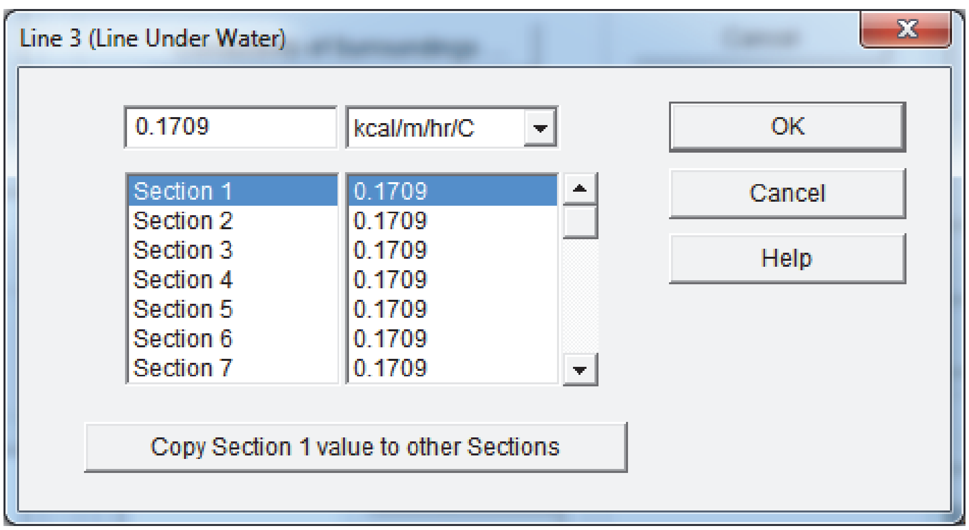


Figure 9: Conductivity of Surrounding (from line6.psd)

Air – Conductivity of Surroundings
This dialog specifies the conductivity of the air surrounding the Line.

Data Item	Description
Air Conductivity	The conductivity of surrounding air at specified intervals.

Water – Conductivity of Surroundings
This dialog specifies the conductivity of the air surrounding the Line.

Data Item	Description
Water Conductivity	The conductivity of surrounding water at specified intervals.

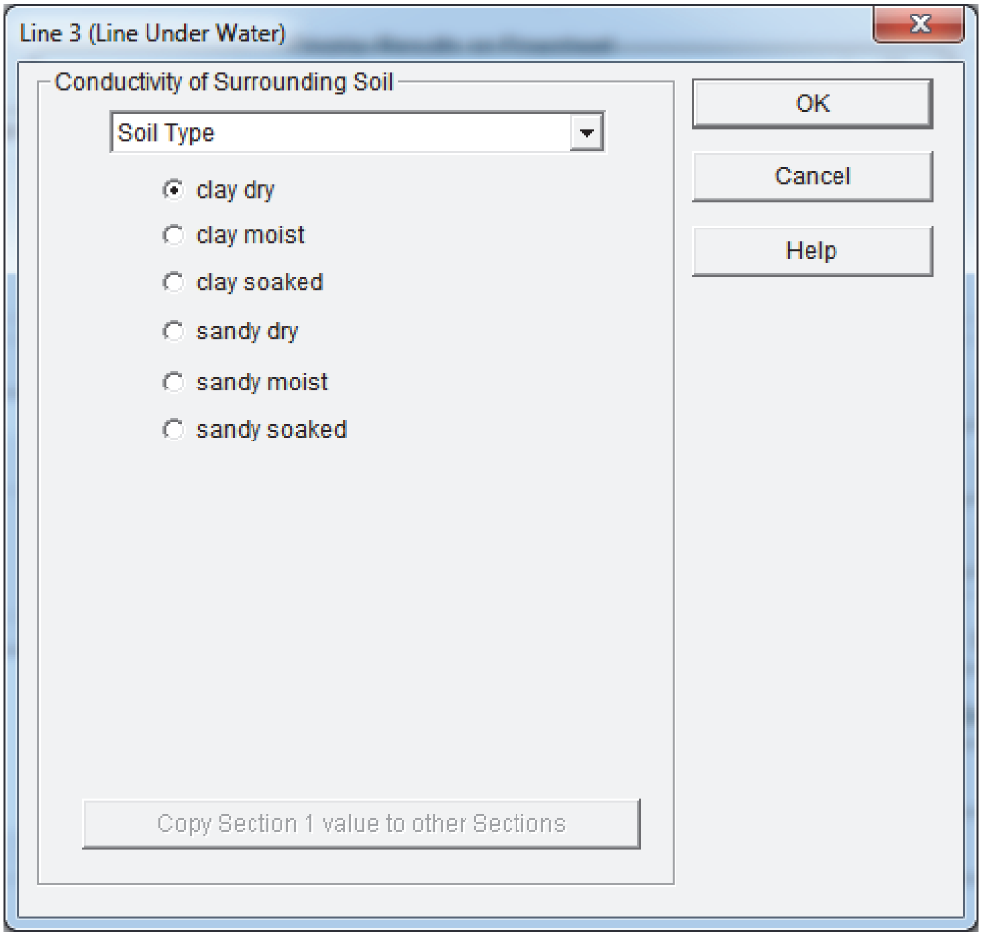


Figure 10: Conductivity of Surrounding (from line6.psd)

Soil – Conductivity of Surrounding Soil

This dialog specifies the conductivity of the soil surrounding the Line. The dialog has a horizontal combo box with two choices:

Soil Type	You may select one of six different types of soil. The conductivity value for each is in the database. This choice is for the entire Line.
Conductivity of Surrounding Soil	You may enter your own conductivity values for each specified intervals along the Line.

Data Item	Description
Soil Type	sandy dry (k = 0.1625 Btu/hr/ft/F), sandy moist (k = 0.275 Btu/hr/ft/F), sandy soaked (k = 0.6 Btu/hr/ft/F), clay dry (k = 0.125 Btu/hr/ft/F), clay moist (k = 0.225 Btu/hr/ft/F), clay soaked (k = 0.375 Btu/hr/ft/F)or
Soil Conductivity	The conductivity of surrounding soil along specified intervals.

Velocity of Surroundings

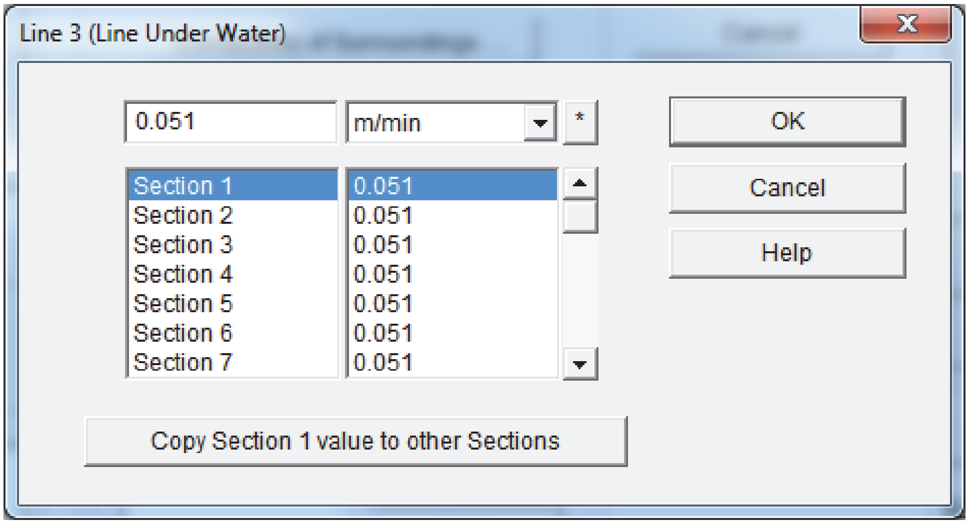


Figure 11: Velocity of Surroundings (from line6.psd)

Air - Velocity of Surroundings

This dialog specifies the velocity of the surroundings (air or water) along the Line.

Data Item	Description
Surroundings Velocity	Specify the velocity of the surroundings for each interval along the line.

Water – Velocity of Surroundings

This dialog specifies the velocity of the surroundings (air or water) along the Line

Data Item	Description
Surroundings Velocity	Specify the velocity of the surroundings for each interval along the line.

Pipe Depth

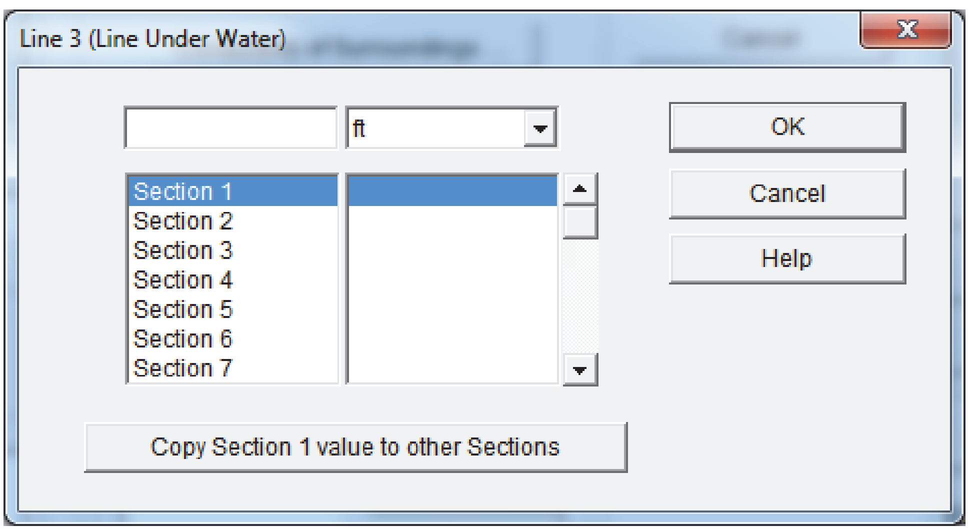


Figure 12: Pipe Depth (from line6.psd)

Pipe Depth

This dialog specifies the depth the pipe is buried at several intervals along the Line.

Data Item	Description
Pipe Depth	The depth of the line at specified intervals.

Pipe Insulation

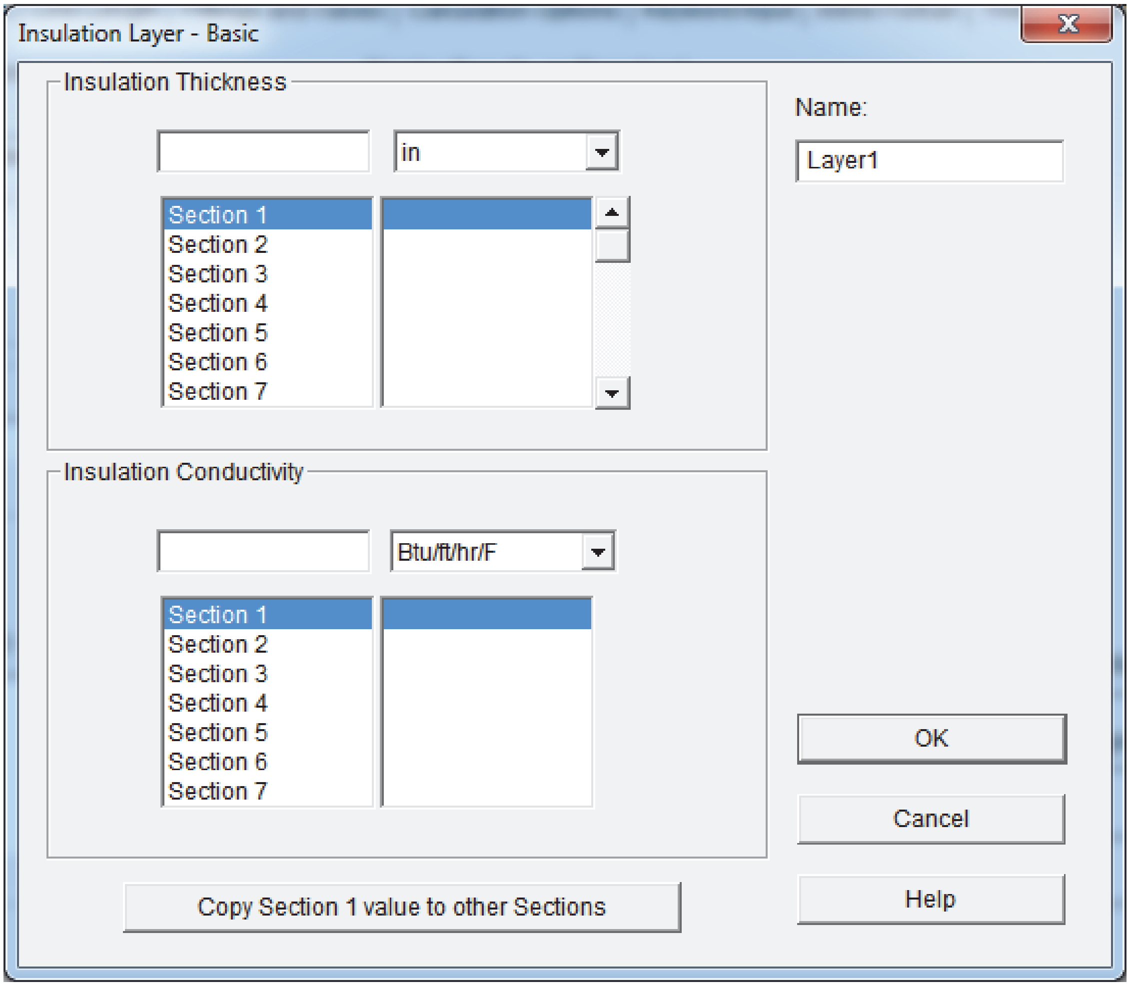


Figure 13: Insulation Layer (from line6.psd)

Pipe Insulation
This dialog has two list boxes for specifications on one layer of insulation- use the top table to specify the Insulation Thickness (in pairs) and the bottom table to specify the Insulation Conductivity (in pairs).

Data Item	Description
Insulation Thickness	Specify the thickness of the layer at each interval.
Insulation Conductivity	Specify the conductivity of the layer at each interval.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click **Load Template** and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the "C-*" before it and enter the units in parenthesis and any values after the =. Any line beginning with "*" will not be written to the DESIGN II input file. However any line with a "C-" (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Line Pressure Drop Examples

There are several sample flowsheets in “Chapter 27: Line Module Samples - c:\designii\samples\equipmnt\line” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.22: LNG Exchanger

General

The LNG Exchanger module is used to specify heating or cooling of up to twenty tubeside streams and twenty shellside streams. Each tubeside stream can have either the same outlet specification or entirely different specifications. Tubeside outlet streams may be recycled back and mixed with a single shellside feed.

Details

Coding Streams:
The order in which all streams are coded for the LNG module is as follows:
1. Recycle tube side streams coded as first input and first output streams.
2. Remaining tube side streams are coded next.
3. Shell side stream(s) are coded as last input and last output stream(s).

Order of Data Entries:
When entering data separated by commas on one command, be sure the data for each tube side stream is entered in the same order as the stream numbers are listed after the Equipment Module Identifier. For example, if the pressure drop for streams 8 and 10 is 5 PSIA, and 10 PSIA for streams 9 and 11, the DEL command is coded as follows:

```
LNG 3 = REFG, 8, 9, 10, 11, 30, -12, -13, -14, -15, -31
DELta pressure TUBe= 5, 10, 5, 10
```

Decide whether you want to use the same type of specification for all tube side streams or use different specification types. Then, select the specification(s) from the appropriate list of commands. If you use different specification types for tube side streams, make sure a specification command is entered for each tube side stream. Each tube side stream is identified by a number n, which is the position of that tube side stream number in the list of stream numbers following the Equipment Module Identifier. For example:

```
LNG 3 = CHIL, 10, 11, 12, 13, 20, -14, -15, -16, -17, -21,
TEMperature OUT 3 = 30
```

The command TEMperature OUT 3 = 30 command specifies the third tube stream, stream number 12, to exit at 30F (as stream number 16). Also, make sure your specification is realistic. Be especially aware of heat transfer constraints. For example, you cannot heat a stream to a temperature higher than that of the hot streams. If you have chosen a specification for the shell side stream(s), you cannot use any specifications for the tube side streams.

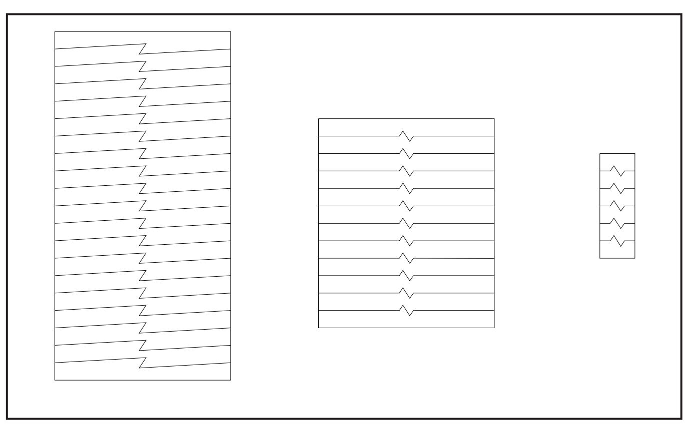
Multiple Shell Side Streams:
If more than one shell side stream is coded, the SHEll STReam command must be entered. Also, no mixing (recycling) of tube side streams to shell side streams is allowed when more than one shell side stream is coded.

Please see the online **DESIGN II** Help topic ***Equipment/LNG Exchanger*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 21: LNG Exchanger*** for more details.

Symbols

The LNG Exchanger unit module has three symbols: LNG Exchanger (up to 3 shell side streams and 5 tube side streams), LNG 11x11 Exchanger (up to 11 shell side streams and 11 tube side streams), and LNG 11x19 Exchanger (up to 11 shell side streams and 19 tube side streams).

The LNG Exchanger module requires that at least two inlet streams and at least two outlet streams be connected to the module.



Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

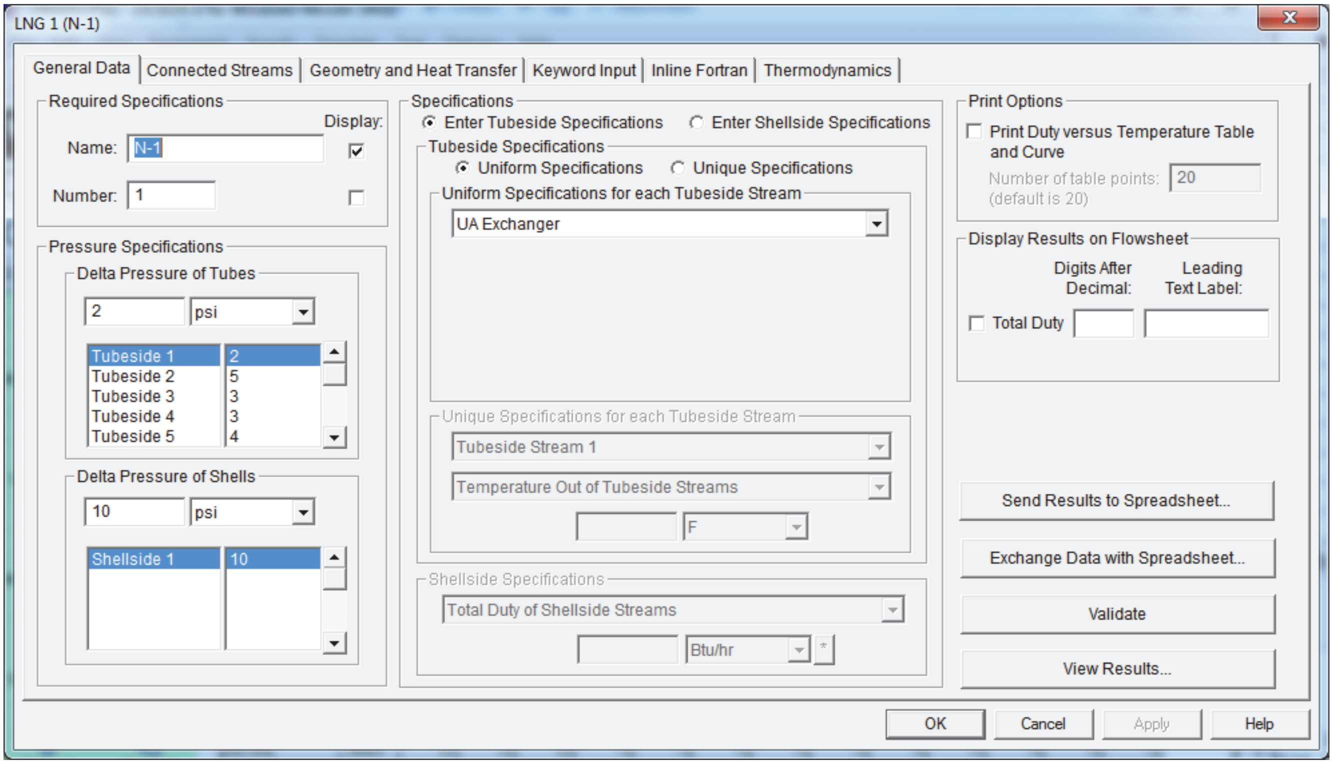


Figure 1: LNG Exchanger Dialog (from lngex1.psd)

General Data

Data Items	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Delta Pressure of Tubes	Click on a tubeside stream from the list ; then enter the pressure drop and select a unit.
Delat Pressure of Shells	Click on a shellside stream from the list ; then enter the pressure drop and select a unit.
Enter Tubeside Specifcations, Specifications/Enter Shellside Specifications	Select what specifications to enter for the LNG exchanger. If you select Tubeside then you can also choose to enter uniform or unique specifications for each tubeside stream.

Tubeside Specifications for Uniform Specifications

Data Items	Description
Temperature Out Tubeside Stream	Enter the estimate or specified temperature of the tubeside outlet stream.
Temperature Approach Tubeside	Enter temperature approach desired for tubeside outlet stream. Default of 5° F.
UA Exchanger	Enter the area and heat transfer coefficient for each tubeside stream (Geometry and Heat Transfer tab).
Duty of Tubeside	Enter the desired enthalpy change or duty available for the tubeside stream.
Temperature Out is Bubble Point	Uses the bubble point.
Temperatue Out is Dew Point	Uses the dew point.

Tubeside Specifications for Unique Specifications

Data Items	Description
Tubeside Stream	Open the list and select the tubeside stream for which you want to set specifications.
Temperature Out of Tubeside	Enter the temperature and select a unit.
Temperature Approach of Tubeside	Enter the approach and select a unit.
Duty of Tubeside	Enter the duty available and select a unit.
Temperature Out is Bubble Point	Uses the bubble point.
Temperatue Out is Dew Point	Uses the dew point.

Shellside Specifications

Data Items	Description
Total Duty of Shellside	Enter the desired enthalpy change or duty available for the shellside stream.
Temperature Out Shellside stream	Enter the estimate or specified temperature of the shellside outlet stream.

Print Options

Print Duty versus Temperature Table and Curve	Select this option to print a Duty versus temperature curves for composite hot and cold streams. Multiple streams for either the shell or tube side will be summed to present a composite hot stream versus a composite cold stream. The results, presented in both tabular and graph form, indicate all critical information such as narrow temperature approach, temperature crossovers, and discontinuity in composite curves. These curves are calculated based on end-point temperature results. Bubble points, dew points, minimum approach temperature, and temperature crosses are marked on the tabular output when applicable.
Display Results on Flowsheet	You can optionally display the following result on the flowsheet: Total Duty If you select to display one or more of these results, enter the number of digits to display to the right of the decimal, along with any text you might want to display before the results

Connected Streams

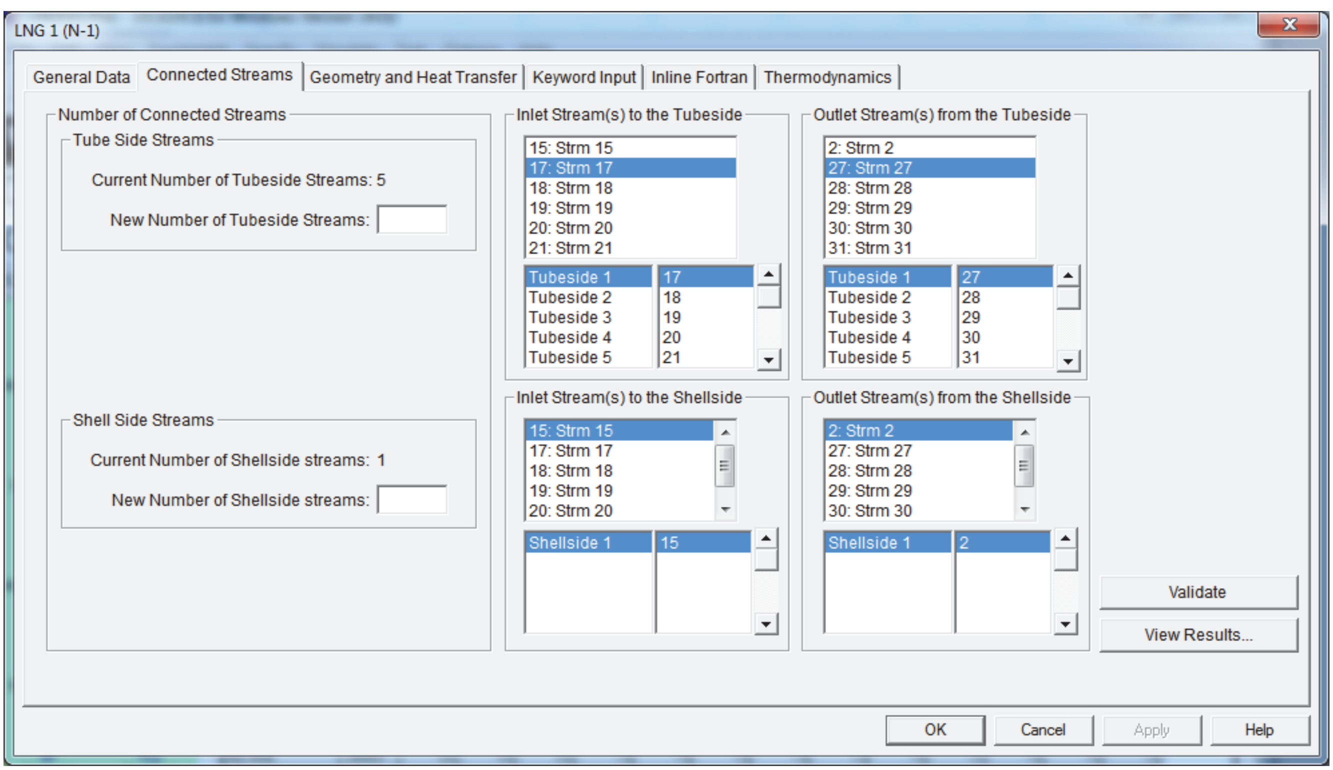


Figure 2: Connected Streams tab on LNG Exchanger Dialog (from lngex1.psd)

Data Items	Description
Number of Tubeside Streams	Enter the number of tube side streams for the LNG Exchanger.
Number of Shellside Streams	Enter the number of shell side streams for the LNG Exchanger.

Tube Side Inlet & Outlet Stream(s)	Identify the stream numbers in and out of the LNG Exchanger connected in the flowsheet for the tube side stream(s).
Shell Side Inlet & Outlet Stream(s)	Identify the stream numbers in and out of the LNG Exchanger connected in the flowsheet for the shell side stream(s).

Geometry and Heat Transfer

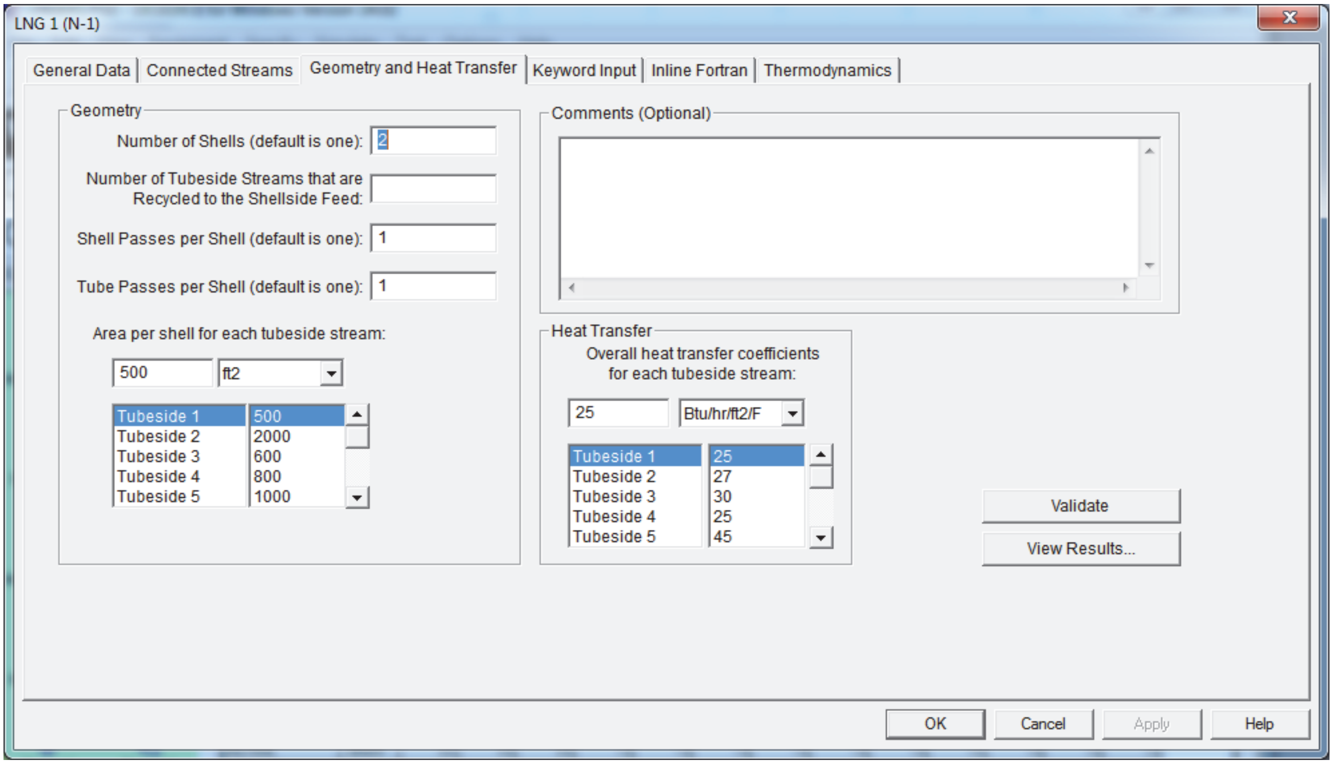


Figure 3: LNG Geometry and Heat Transfer Dialog (from lngex1.psd)

Data Items	Description
Number of Shells	Enter the number of shells (default is 1).
Number of Tubeside Streams	Enter the number of tubeside streams that are recycled to the shell side using a mixer.
Shell Passes per Shell	Enter the number of shell passes per shell (default is 1).
Tube Passes per Shell	Enter the number of tube passes per shell (default is 1).
Area per Shell for each Tubeside Stream	Enter the area per shell for each tubeside stream.
Heat Transfer Coefficient	Enter the heat transfer coefficent for each tubeside stream.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II Reference Guide.

LNG Exchanger Examples

There are several sample flowsheets in "Chapter 28: LNG Exchanger Samples - c:\designii\samples\equipmnt\lngexc" of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.23: Mass Balance

General

The Mass Balance module is used for setting the flowrate of a recycle loop at a certain point and / or calculating the required makeup flowrates for certain components for a recycle loop. An essentially unlimited number of input streams can be coded; they will be adiabatically mixed to the lowest feed stream pressure. Only one outlet stream is allowed for the module.

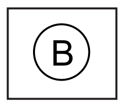
Details

The components entering and leaving the flowsheet can be balanced & a specific feed stream can be assigned as a make-up stream. The feed stream to be varied by the makeup calculations of Mass Balance module can be attached to the Mass Balance module or any other module in the flowsheet. These are useful in amine contacting and glycol dehydration simulations.

Please see the online **DESIGN II** Help topic ***Equipment/Mass Balance*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 22: Mass Balance*** for more details.

Symbols

The Mass Balance unit module has one symbol:



The Mass Balance module requires that you connect one or more inlet streams and one outlet stream to the module.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

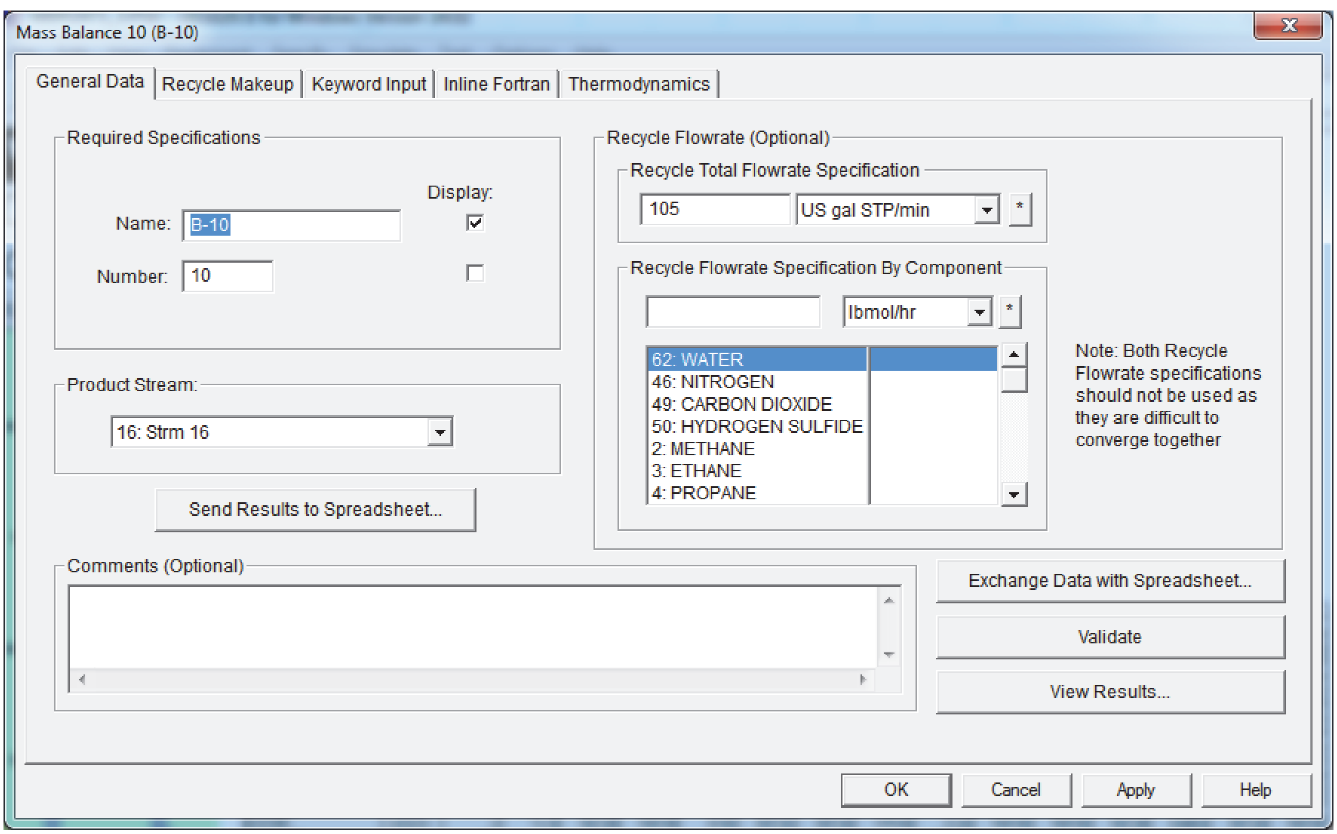


Figure 1: Mass Balance Properties Dialog (from mxpInt1.psd)

General Data

Data Items	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Product Stream	Open the drop down list and select the stream to use as the product stream.
Recycle Total Flowrate Specification	Enter the flowrate or flowrates to be set for the output stream. This command is optional. There is no default value. You should not set both the recycle total flowrate and the recycle flowrate by component as they are difficult to converge together.
Recycle Flowrate Specifcation by Component	Click on a component name from the list then enter a recycle flowrate for the component for the stream. This command is optional. You can either enter a value here or use the Recycle output Total Flowrate Specification option. You cannot use both; they are difficult to converge together. To enter base quantity and time units, click the *.

Optional Specification: Recycle Makeup

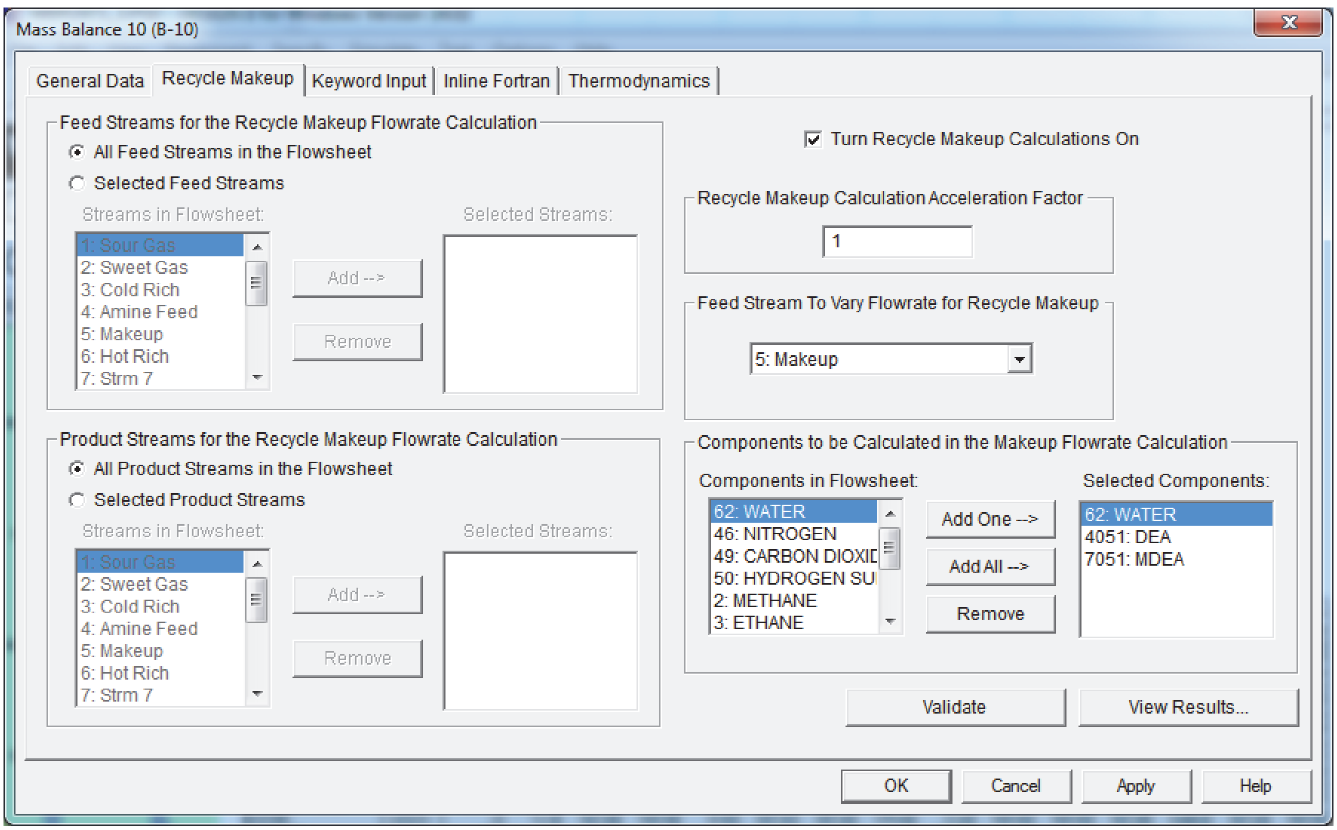


Figure 2: Recycle Makeup Dialog (from mxplnt1.psd)

Turn Recycle Makeup Calculations On
Use this checkbox to turn on the recycle makeup calculations.

Feed Streams for the Recycle Makeup Calculation
Use this group box to specify the feed streams for the recycle makeup calculation. You can either select all feed streams for the flowsheet or selected feed streams for the flowsheet. If you choose selected feed streams for the flowsheet then you must choose at least one feed stream in the flowsheet.

Product Streams for the Recycle Makeup Calculation
Use this group box to specify the product streams for the recycle makeup calculation. You can either select all product streams for the flowsheet or selected product streams for the flowsheet. If you choose selected product streams for the flowsheet then you must choose at least one product stream in the flowsheet.

Recycle Makeup Calculation Acceleration Factor
Enter the acceleration factor to be used for calculating the delta change for the recycle makeup flowrate. The default is 1.0. A typical value for this command might be in the range 0.5 to 1.0, usually 1.0 or 0.8. This command is optional.

Feed Stream to Vary Flowrate for the Recycle Makeup Calculation
Use this group box to specify the feed stream for its flowrate to be varied with the calculated recycle makeup flowrate. The feed stream to be varied must be selected in order to calculate the recycle makeup flowrate. This stream does not have to be connected to the mass balance module.

Components to be Calculated in the Recycle Makeup Calculation
Select the components to be included in the recycle makeup flowrate calculation. Only components that are being lost to the recycle (like water, MEA, DEA, MDEA, etc…) should be part of the makeup flowrate calculation. Components must be selected in order to calculate the recycle makeup flowrate.

Required Specifications: Keyword Input

Click the **Keyword Input** tab. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II Reference Guide.

Mass Balance Examples

There are several sample flowsheets in “Chapter 29: Mass Balance Samples - c:\designii\samples\equipmnt\masbal” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.24: Mixer

General

The Mixer will mix multiple input streams and adiabatically flash the mixture to the lowest inlet stream pressure.

Details

The Mixer is a single-purpose module that is used to mix multiple inlet streams. The feed streams are adiabatically flashed to the lowest inlet stream pressure. Since most DESIGN II Equipment Modules accept more than one inlet stream and adiabatically mix them before calculation, the Mixer is seldom required for flowsheet simulation.

The advantage of using the Mixer with a single outlet steam is that the properties of the mixed stream are reported in the printout. This is not the case in modules that automatically mix multiple feeds ; then proceed with calculations.

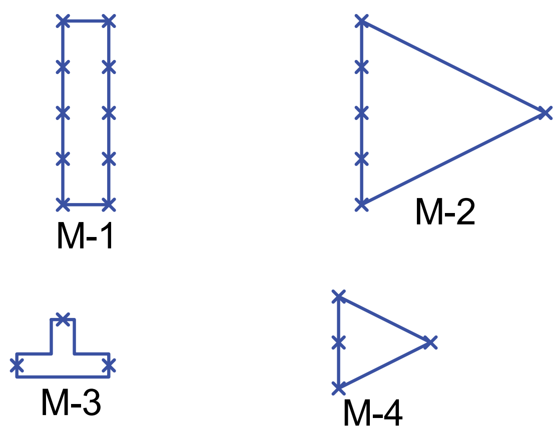
If you are not interested in the mixed outlet stream, and are using one of the following Equipment Modules, you can eliminate the Mixer:

Distillation	Line	Refine	Fired Heater	Reactor	Shortcut Fractionator

Please see the online **DESIGN II** Help, topic ***Equipment/Mixer*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 23: Mixer*** for more details.

Symbols

The Mixer unit module has four symbols:



The Mixer module requires that one inlet and one outlet stream be connected to the module. Up to nine inlet streams may be connected to this module.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

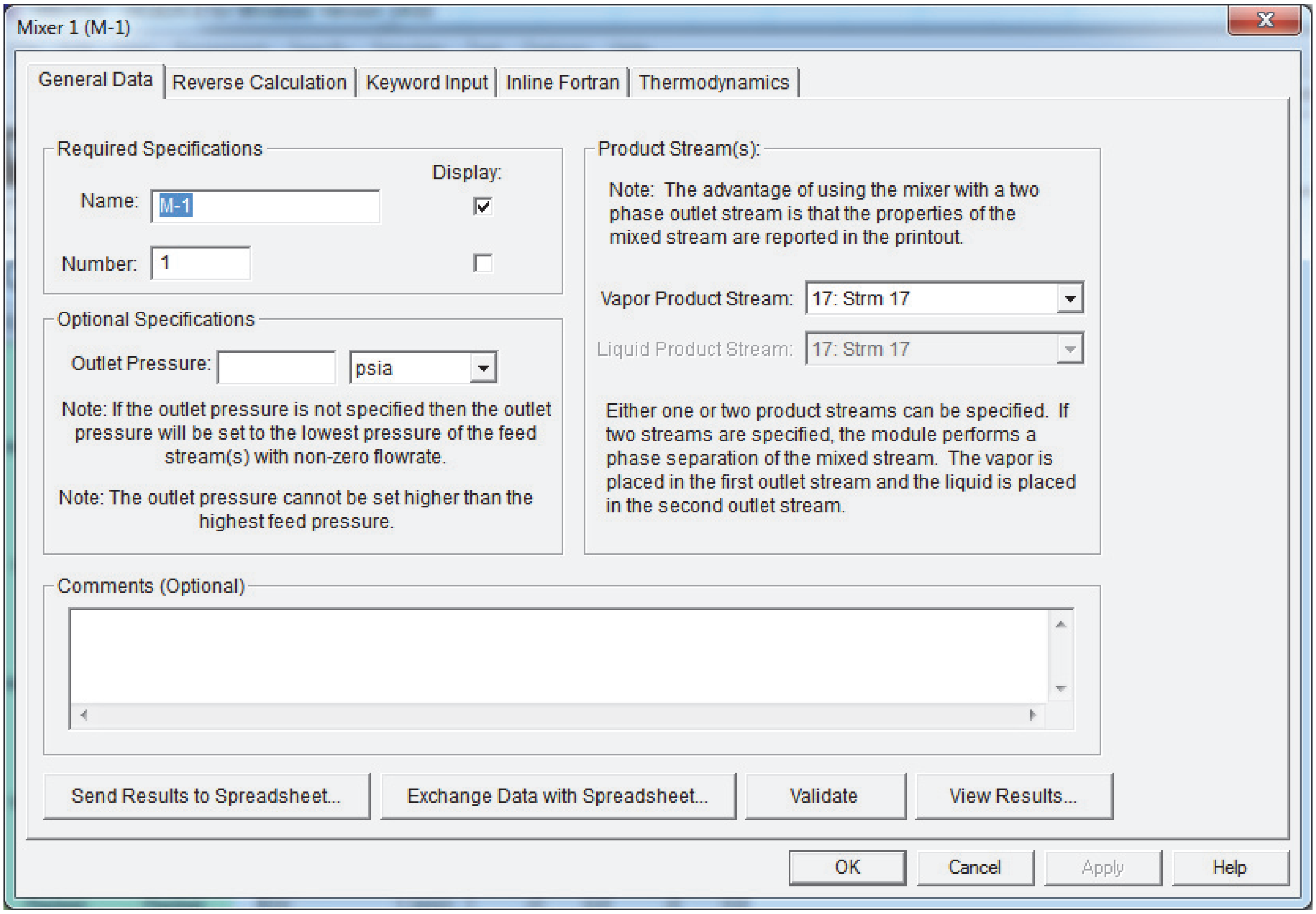


Figure 1: Mixer Dialog (from Mix.psd)

This dialog is used to enter the basic specifications for the Mixer Module.

Data Items	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Outlet Pressure	No specifications are required for the Mixer. A specification for the Outlet Pressure is allowed. If the outlet pressure is not specified then the outlet pressure will be set to the lowest pressure of the feed stream(s) with non-zero flowrate.

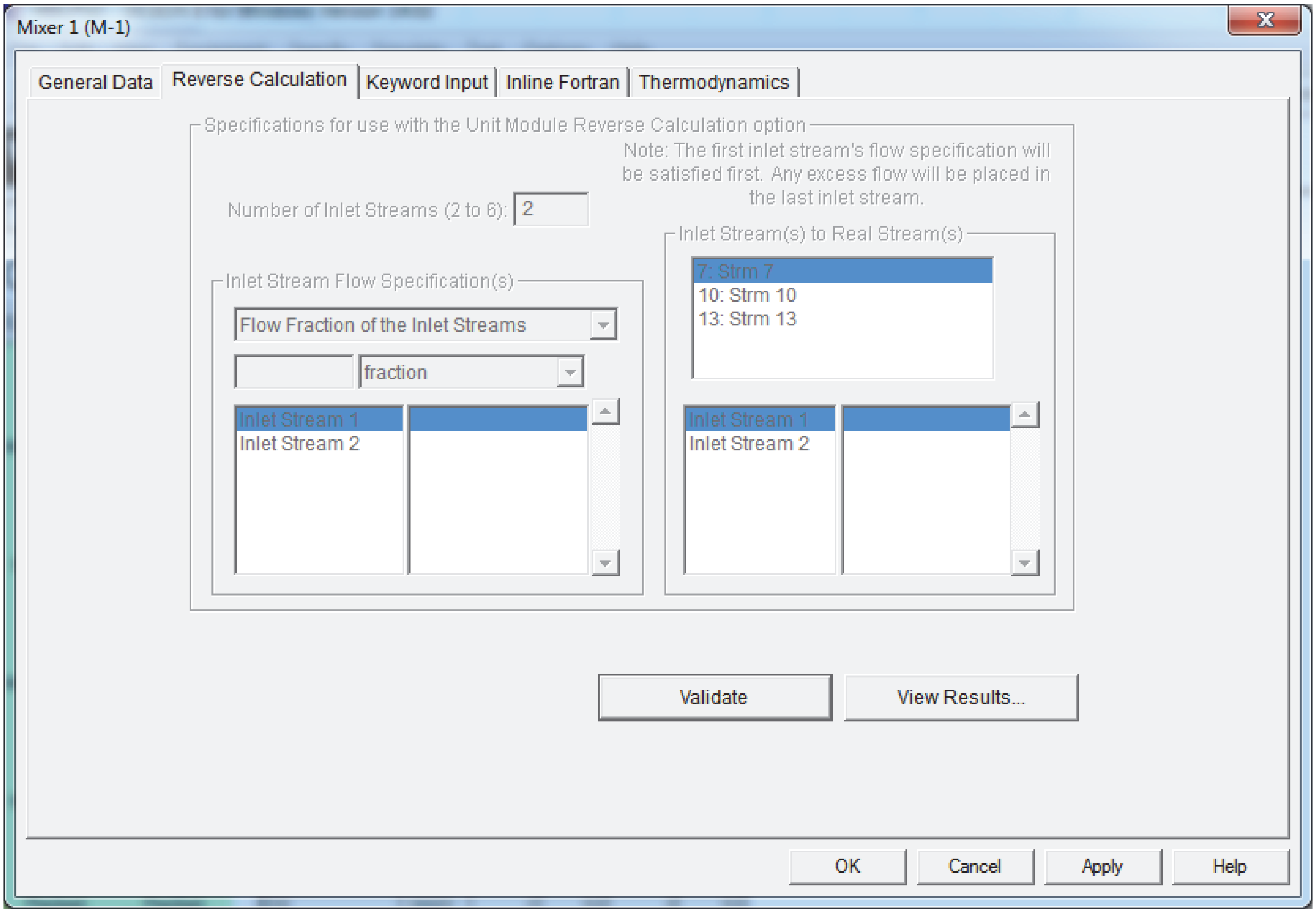


Figure 2: Mixer Dialog (from Mix.psd)

Mixer, Mixer 2, 3

The mixer will combine multiple input streams and adiabatically flash the mixture to the lowest inlet stream pressure.

The only difference between the mixers is the symbol used on the flowsheet.

For more information about the mixer, select Detail.

The mixer requires that one inlet and one outlet stream be connected to it. Up to nine inlet streams can be connected to this module.

Optional Specifications: Keyword Input

Click Keyword Input. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Mixer Examples

There are several sample flowsheets in “Chapter 30: Mixer Samples - c:\designii\samples\equipmnt\mixer” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.25: Multiple Flashes

General

The Multiple Flashes module can perform isothermal, adiabatic, isentropic, liquid fraction (molar or mass), bubble point, dew point, and water dew point calculations. Either temperature or pressure can be specified for the above options. Heat addition/removal can be included for adiabatic flash calculations. An isothermal flash is the default option.

One inlet stream can be flashed up to fifty times or up to fifty separate inlet streams can be flashed. No phase separation will be performed, but the phase information will be contained in the detailed stream summaries. If phase separation is required for any outlet stream, use a VALve or FLAsh module.

Details

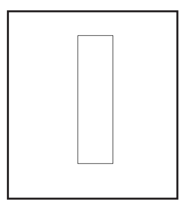
Two options are available for stream input in the Multiple Flashes module. You can have one inlet stream and up to 50 outlet streams (all same composition and flow as inlet stream) or up to fifty inlet streams and an equal number of outlet streams. Specifications and calculation options always refer to outlet stream numbers, not inlet stream numbers.

The default calculation option is an isothermal flash. If no TEMperature OUT and/or PREssure OUT command is used, the flash will be performed at the inlet temperature and pressure for the stream. For all flashes other than isothermal, you can specify either the temperature or the pressure out. The program will then solve for the corresponding pressure or temperature. If both TEMperature OUT and PREssure OUT are specified, PREssure OUT is treated as the specification and the TEMperature OUT command will be ignored.

Please see the online **DESIGN II** Help, topic ***Equipment/Multiple Flashes*** or the **DESIGN II Unit Module** Reference ***Guide Chapter 24: Multiple Flashes*** for more details.

Symbols

The Multiple Flashes unit module has one symbol:



The Multiple Flashes module requires that at least one inlet stream and at least one outlet stream be connected to the module.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

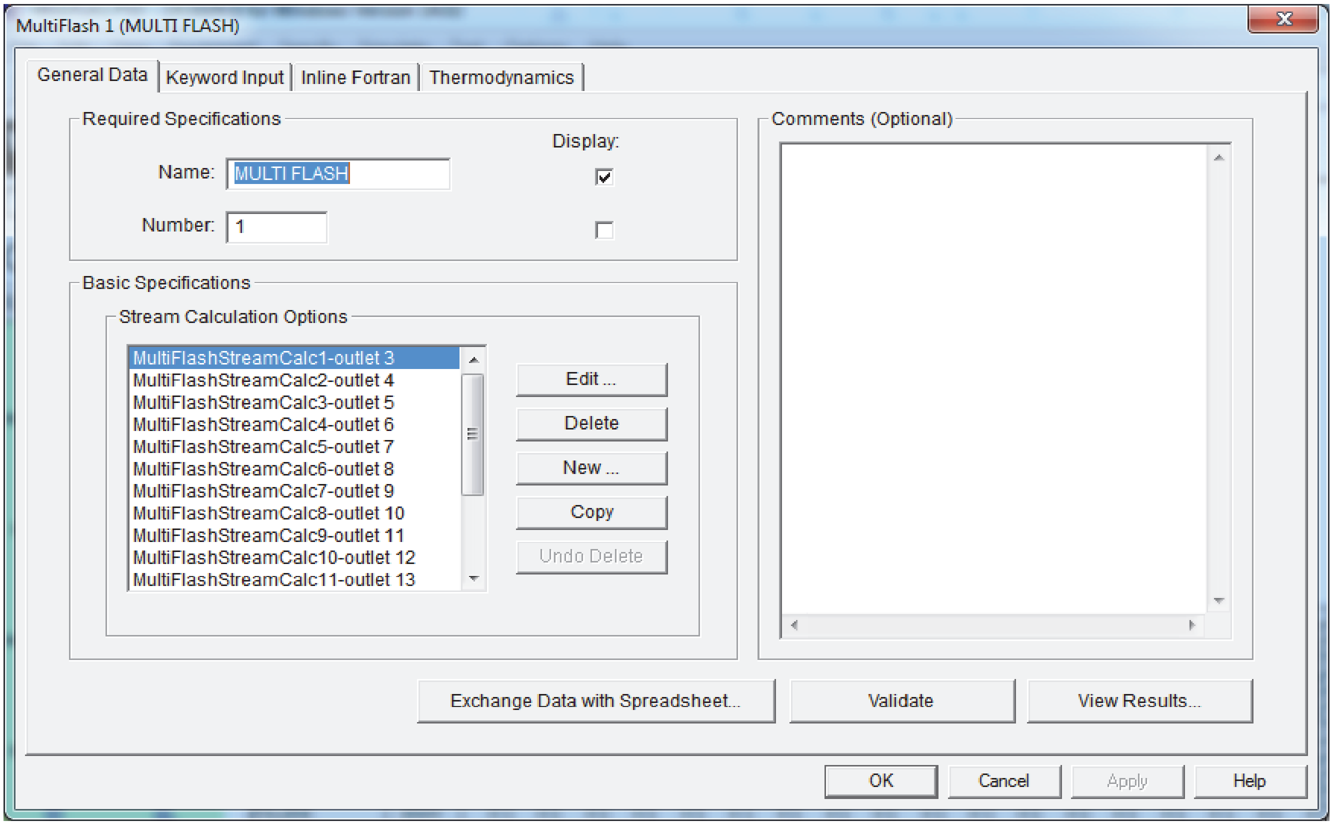


Figure 1: Multiple Flashes Dialog (from mulfla1.psd)

Multiple Flash Stream Calculations

The default calculation option for any outlet stream is an isothermal flash. Other available calculation options are: Adiabatic, Heat added, Isentropic, Bubble Point, Dew Point, Dew Water Liquid Fraction (molar & mass), Vapor and Liquid.

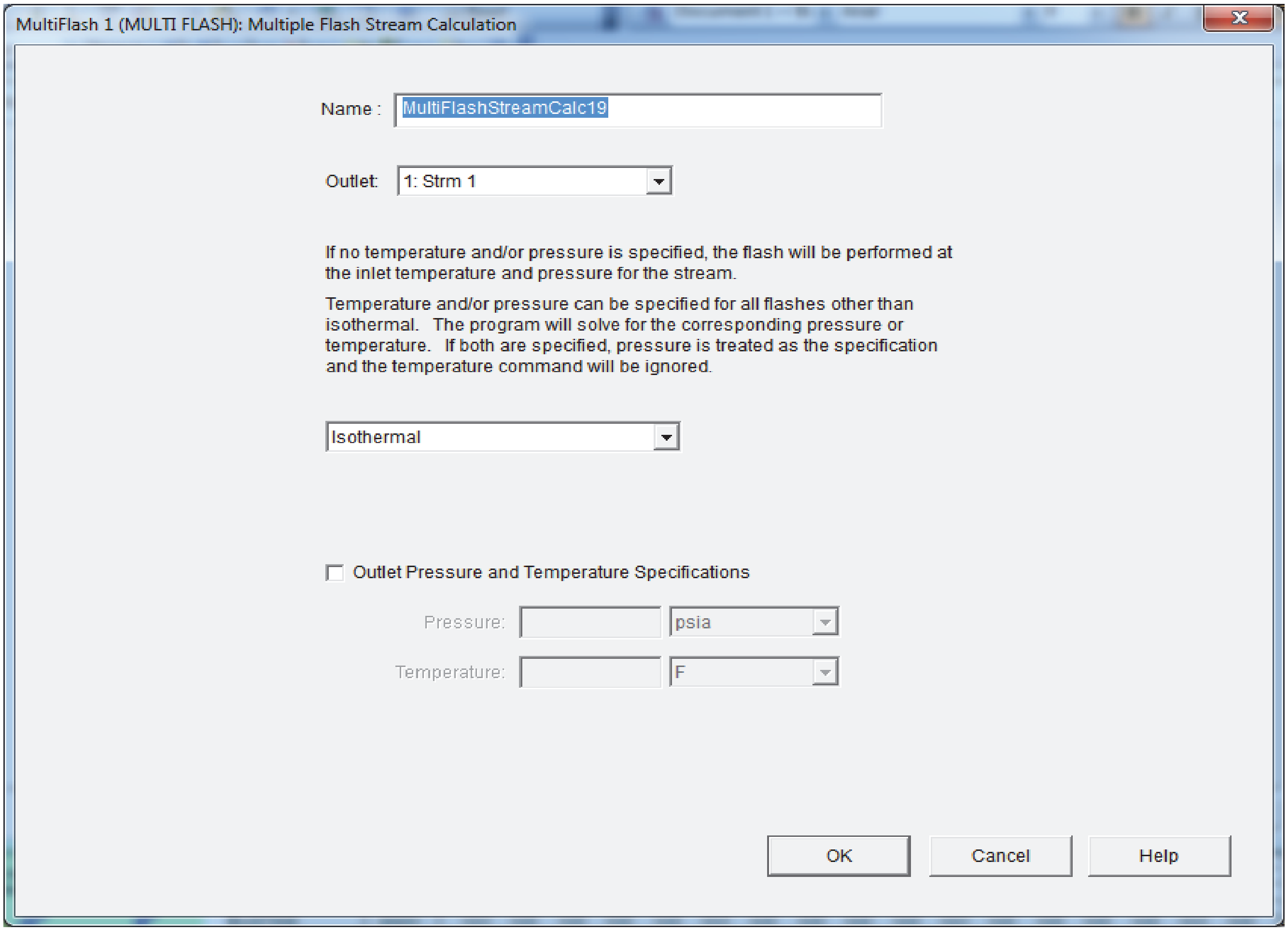


Figure 2: Multiple Flash Stream Calculation Dialog (from mulfla1.psd)

Data Item	Description
Isothermal	Enter the outlet stream that is to be flashed isothermally.
Adiabatic	Enter the outlet stream that is to be flashed adiabatically.
Heat Added	Enter the outlet stream and the amount(s) of heat added to or subtracted from the flash. A negative value indicates heat subtracted from the flash.
Isentropic	Enter the stream numbers for which an isentropic flash is to be performed.
Bubble Point	Enter the outlet stream number for which bubble point is to be calculated.
Dew Point	Enter the outlet stream number for which dew points is to be calculated (hydrocarbon dew point for immiscible flash).
Dew Water	Enter the outlet stream number for which water dew point is to be calculated (Water has to be treated as immiscible).
Liquid Fraction Molar	Enter the outlet stream number and the desired molar liquid fraction.
Liquid Fraction Mass	Enter the outlet stream number and the desired mass liquid fraction.
Vapor	Enter the outlet stream number set to be the vapor phase at the inlet temperature and pressure (or at a specified temperature and pressure).
Liquid	Enter the outlet stream number set to be the liquid phase at the inlet temperature and pressure (or at a specified temperature and pressure).

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. In future releases of the product, these instructions will be replaced with fill-in-the-blank dialog boxes. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*”

will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Multiple Flashes Examples

There are several sample flowsheets in “Chapter 31: Multiple Flashes Samples - c:\designii\samples\equipmnt\mulflash” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.26: Multiple Phase Flashes

General

The Multiple Phase flash module can be used to rigorously separate multiple phase streams. One inlet and at least three outlet streams should be coded. The vapor phase is placed in the first outlet stream and the liquid phases are placed in the other outlet streams.

Details

The module can be used with activity coefficient K-value options such as RENon, UNIQUAC, and UNIFAC or cubic equations of state such as PENg-Robinson K, SOAVEK, APISOAVEK, MODified PENg-Robinson K, or Soave-Kabadi-Danner (SKDK). Vapor-liquid and/or liquid-liquid equilibrium data can be fitted using RENon, UNIQUAC or an equation of state. Alternately, the UNIFAC option can be used to estimate infinite dilution activity coefficients. To get the best results for a three-phase flash, you should fit either liquid-liquid equilibrium data or a combination of vapor-liquid and liquid-liquid equilibrium data to either the RENon or UNIQUAC equations.

The three-phase system parameters for these correlations are typically temperature dependent. Better results can be expected if the degree of temperature extrapolation is low. If possible, the data which is being fitted should be at operating conditions or span the operating conditions. If the data covers a wide temperature range, the temperature dependency parameter (B12) should be varied during the data regression.

NOTE: If the value of the non-randomness parameter (C12) for the RENon equation exceeds 0.426, no liquid-liquid phase splitting will be predicted.

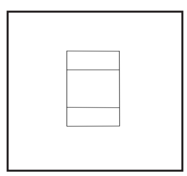
The UNIFAC group contribution technique can also be used, but it should not be considered as a substitute for experimental data. The WILson equation should not be used for these calculations as it has no provision for predicting behavior in the partially miscible region.

An estimate of the outlet stream's phase and flowrate can be entered in the GENeral section to provide a starting point for the MULtiple PHAse flash calculation. The standard two-phase stream initialization will be used as a default.

Please see the online **DESIGN II** Help, topic ***Equipment/Multiple Phase Flash*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 25: Multiple Phase Flash*** for more details.

Symbols

The Multiple Phase Flashes unit module has one symbol:



The Multiple Phase Flash module requires that at least one inlet stream and at least one outlet stream be connected to the module.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

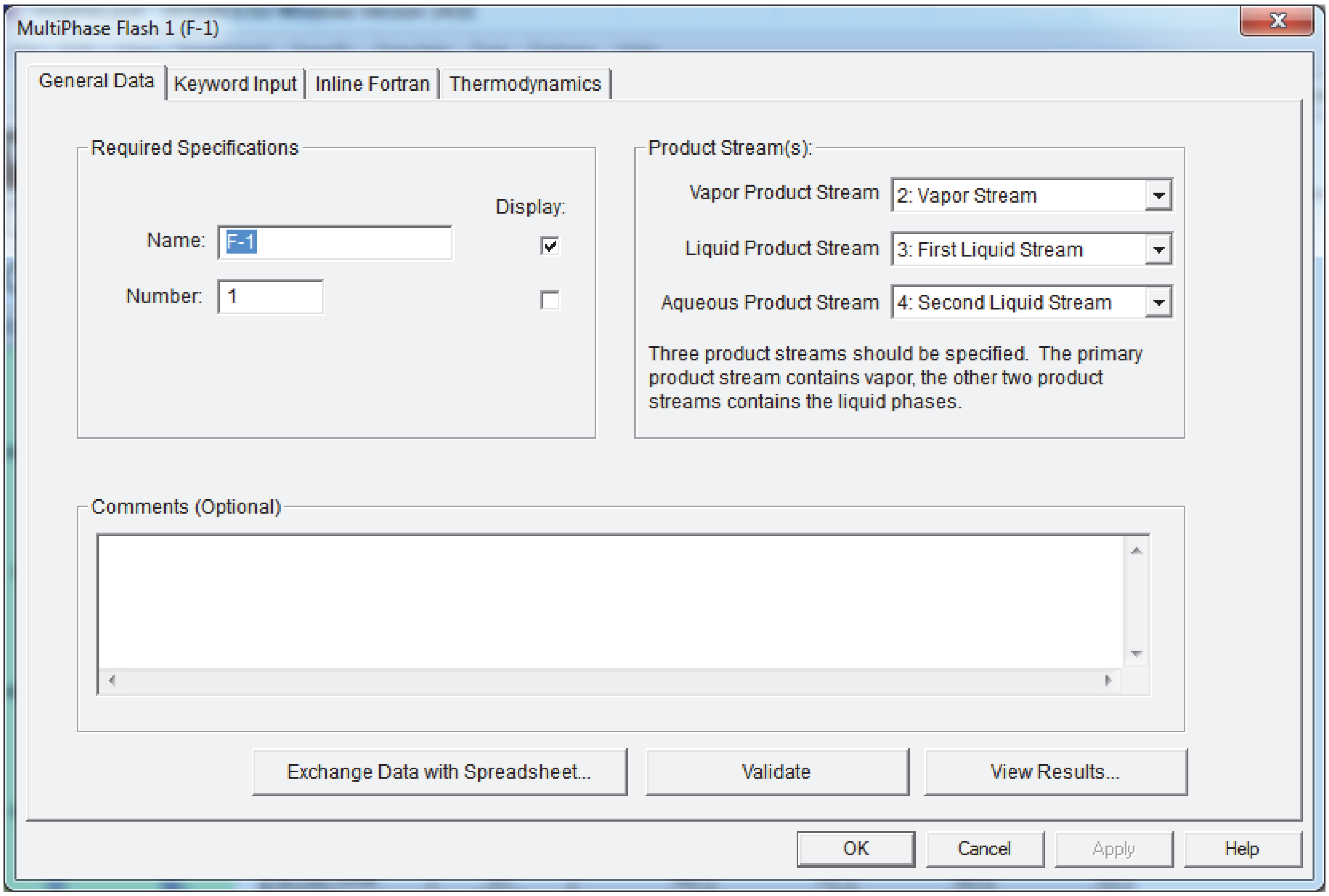


Figure 1: Multiple Phase Flashes Dialog (from mulpha1.psd)

Product Streams	Select a Vapor Product, Liquid Hydrocarbon Product Stream, and/or Aqueous Product Stream from the drop down list. If two product streams are specified, the vapor is placed in the primary product stream and liquid phased are placed in the other two streams.

Required Specifications: Keyword Input

Click the **Keyword Input** tab. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the "C-*" before it and enter the units in parenthesis and any values after the =. Any line beginning with "*" will not be written to the DESIGN II input file. However any line with a "C-" (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Multiple Phase Flash Examples

There are several sample flowsheets in “Chapter 32: Multiple Phase Flash Samples - c:\designii\samples\equipmnt\mulphase” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.27: Packed Absorber Column

General

The packed absorber has preliminary calculations for a packed column by using the number of sections, Height Equivalent of a Theoretical Plate (HETP) per section, height per section and pressure drop per unit height per section.

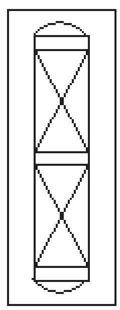
Details

The Packed Colummn module is used for rigorous simulation of absorbers, fractionators, strippers and other types of single-column configurations. Most single-column units can be modeled, including demethanizers, stabilizers, and sour water strippers. This module can be used for design, modification, or optimization of rigorous packed columns. Extensive thermodynamic options are available. With the use of the proper option, the operation of existing units can be duplicated within several percent of actual plant data.

Please see the online **DESIGN II** Help, topic ***Equipment/Packed Column*** or the **DESIGN II Unit Module** Reference ***Guide Packed Column*** for more details.

Symbols

The Packed Column unit module has one symbol:



The packed absorber requires that one inlet stream and one outlet stream be connected to it.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

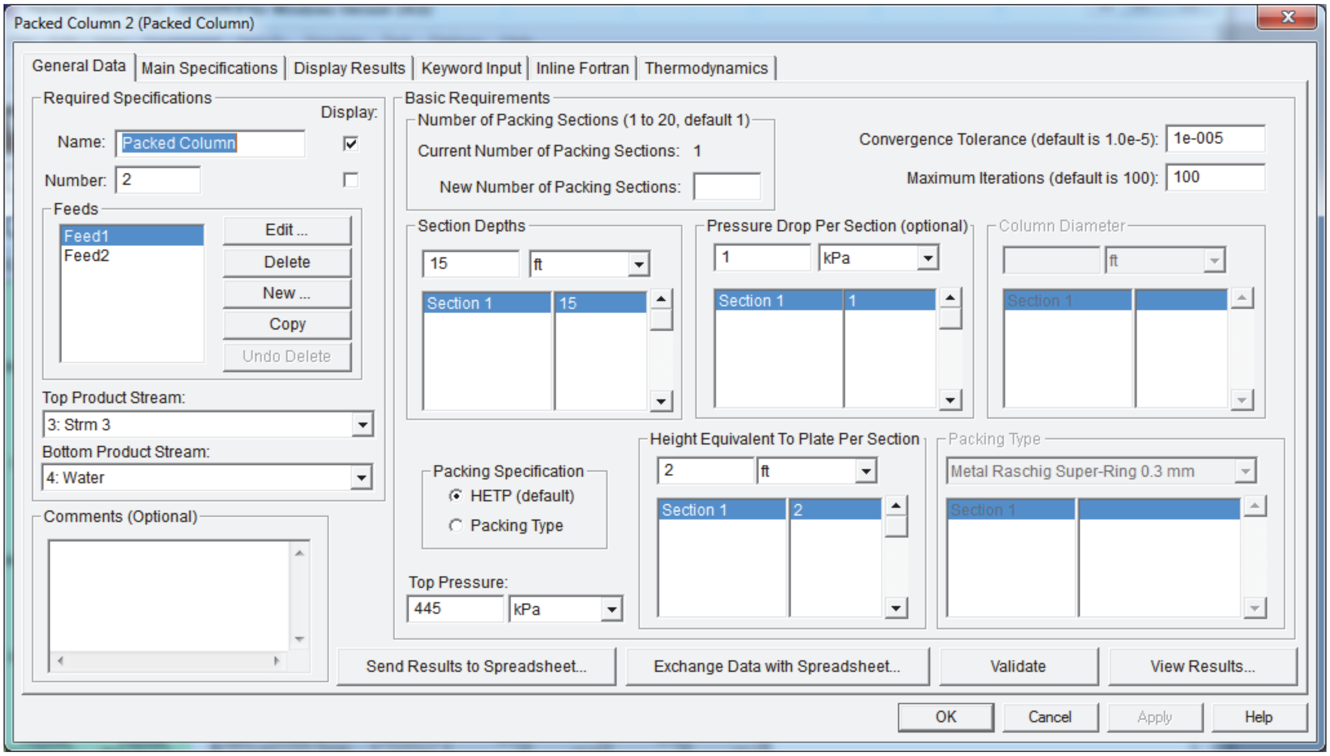


Figure 1: Packed Column Dialog (from Packed Column.psd)

General Data

Click the **Keyword Input** tab. Click Load Template and follow the instructions to specify this eqThis dialog is used to enter the basic specifications for the Mixer Module.

Data Items	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Top Product Stream	Open the drop down list and choose a stream to use as the top product.
Bottom Product Stream	Open the drop down list and choose a stream to use as the top product.
Feeds	Enter feeds to use. Click the New button to display the Feed - Specifications dialog that you can use to create new feed specifications. Select an existing feed and click the Edit button to modify the feed specifications. Select an existing feed and click the Delete button to remove that feed. Select an existing feed and click the Copy button to replicate the feed. You can then use the Edit function to make any minor changes to the copied feed.
Top/Bottom Product Streams	Open the drop down lists and select the desired stream to use for the top or bottom product stream, respectively.
Display Results on Flowsheet	You can optionally display the following results on the flowsheet: Number of Sections Top Temperature Bottom Temperature If you select to display one or more of these results, enter the number of digits to display to the right of the decimal.
New Number of Packing Sections	View the total packing sections. Enter a value (between 1 and 20) to set the total number of packing sections.
Section Depth	Click on a section from the list and enter the depth of the section then select a unit.
Packing Specification	Select HETP (the default) or Packing Type.

Top Pressure	Enter the packed column's top pressure. You must enter a value.
Pressure Drop Per Section	Click on a section from the list then enter the drop in pressure per section and select a unit.
Column Diameter	If you selected Packing Type, select a section and enter the column diameter for each section and select a unit.
Height Equivalent to Plate	If you select HETP as the packing specification, click on a section from the list then enter a Height Equivalent of a Theoretical Plate (HETP) and select a unit. This must be less than section depth; if the HETP is greater than section depth then the number of stages is set to one.
Packing Type	If you selected Packing Type, select a section ; then select the packing type from the top down list.
Convergence Tolerance	Enter the maximum acceptable tolerance for the column. The default is 0.1 E-5.
Maximum Iterations	Enter the maximum number of matrix inversions for column solution.

Required Specifications: Feed

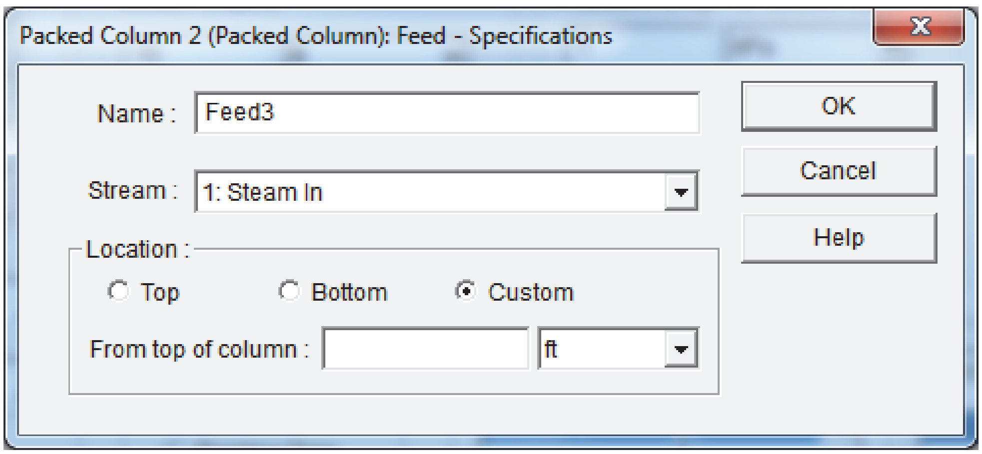


Figure 2: Feed Dialog (from Packed Column.psd)

Data Item	Description
Name	View the name of the feed or change it.
Location from Top of Column	Enter the location for the feed from the top of the column and select a unit.
Stream	Select the flowsheet stream, from the drop-down list, that represents the feed.

Required Specifications: Main Specifications

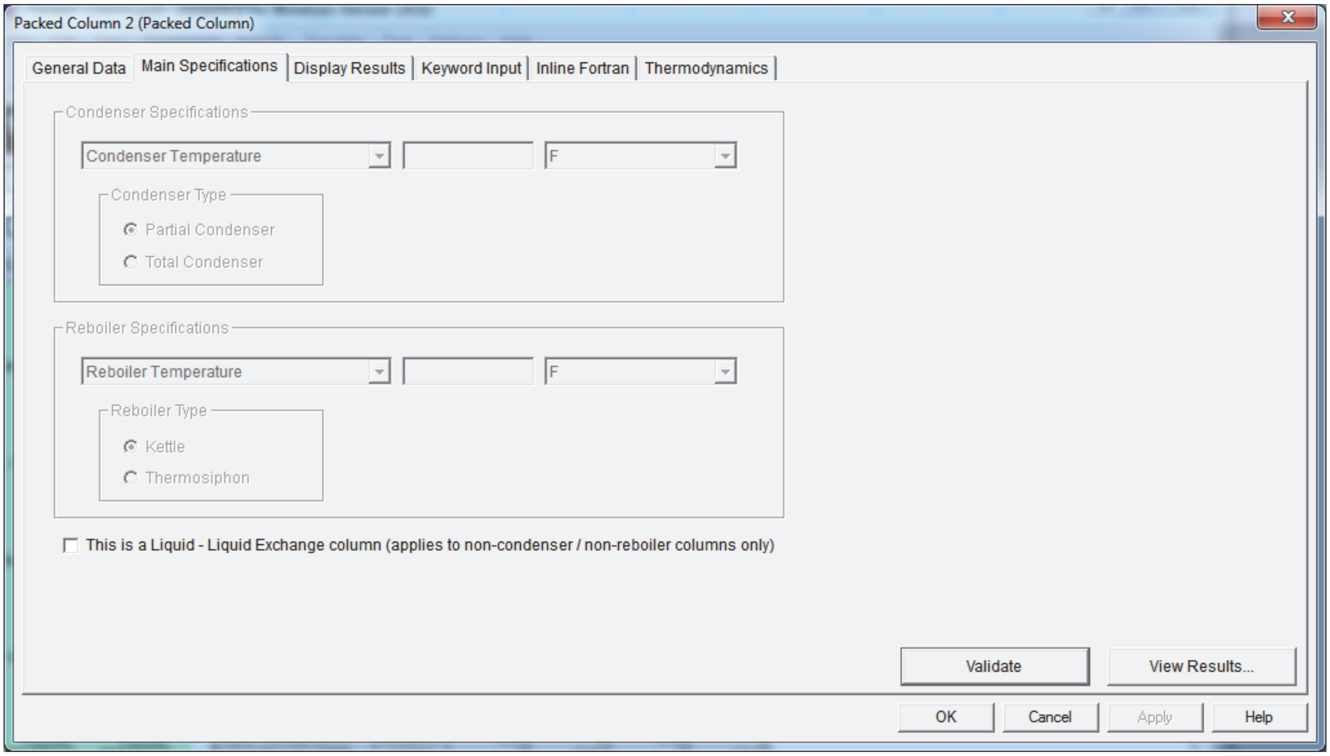


Figure 3: Main Specifications tab (from Packed Column.psd)

Data Item	Description
Condenser Specifications	Select the desired condenser specification from the list and enter the relevant value. You can choose temperature or duty. You should use the Super or Super Plus convergence option with the duty specification. Click on the condenser type, either Partial or Total.
Reboiler Specifications	Select the desired reboiler specification from the list and enter the relevant value. You can choose temperature or duty. For temperature, enter the reboiler outlet temperature. Except for reboiled absorbers, you should use the Super or Super Plus convergence option with this specificationClick on the reboiler type, either Kettle or Thermosiphon.
This is a Liquid-Liquid Exchange Column	Select this checkbox to indicate that the distillation column is a liquid-liquid exchange column; this option only applies to non-condenser or non-reboiler columns.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Chapter 6.28: Phase Envelope

General

The Phase Envelope module calculates dew and bubble points for specified pressures. The properties of the feed stream are passed to the output stream. Critical points are rigorously calculated using the Peng-Robinson equation of state. Binary interaction parameters will be considered if they are present. See Calculation Options: Stream (Mixture) Properties section for critical pressure and temperature calculations for natural gas streams.

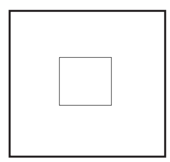
Details

Once the critical pressure is reached (normally on the dew point side of the envelope) further dew point calculations would be meaningless. DESIGN II will attempt to calculate bubble points to fill in the points at the top of the envelope. Similarly, if critical is on the bubble point side, DESIGN II will attempt to calculate dew points to fill in the top of the envelope.

Please see the online **DESIGN II** Help, topic **Equipment/Phase Envelope** or the **DESIGN II** Unit Module Reference Guide ***Chapter 26: Phase Envelope*** for more details.

Symbols

The Phase Envelope unit module has one symbol:



The Phase Envelope module requires that at least one inlet stream and at least one outlet stream be connected to the module.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

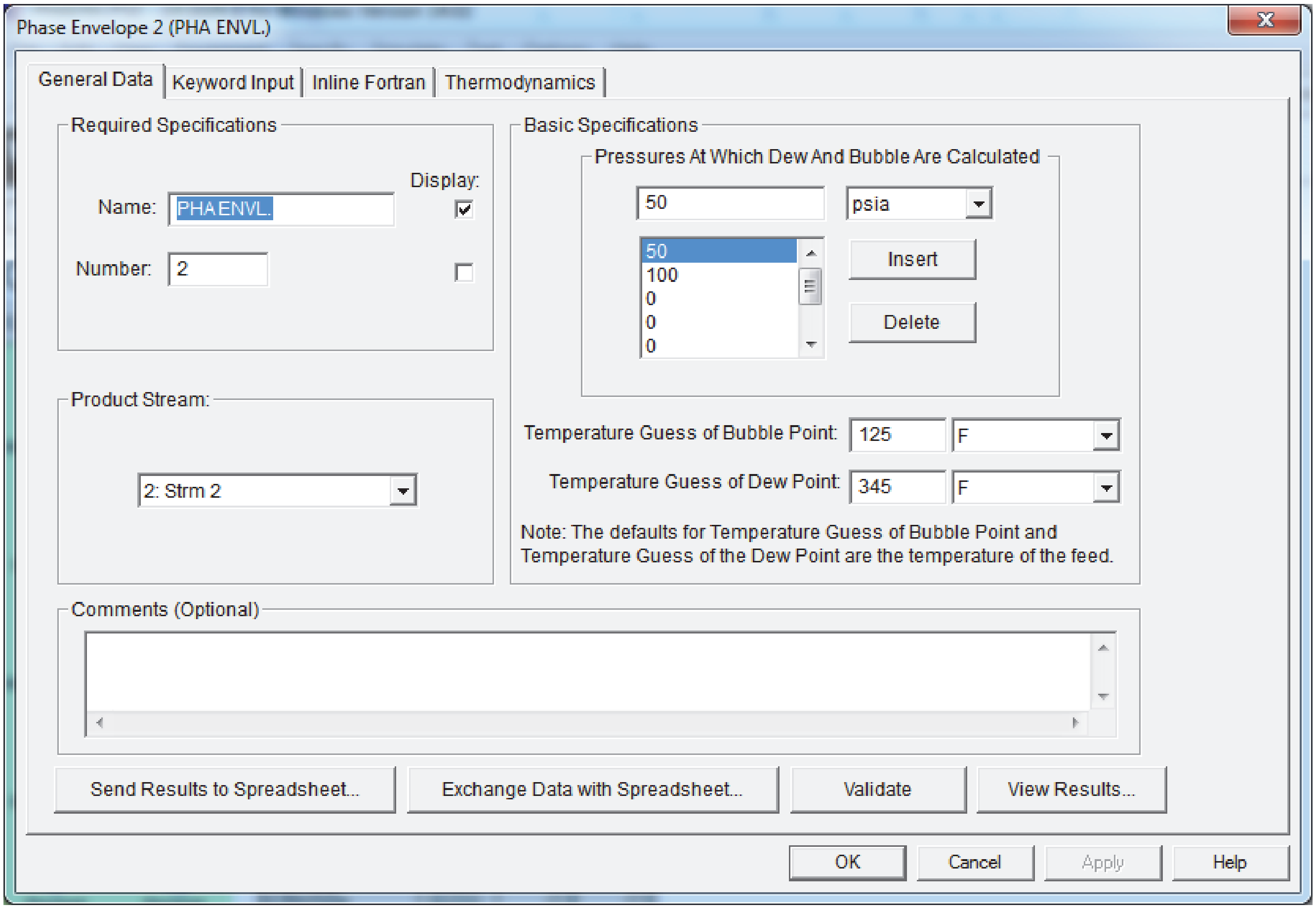


Figure 1: Phase Envelope Dialog (from phaenv.psd)

General Data

This dialog is used to enter the basic specifications for the Phase Envelope Module.

Data Items	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Product Stream	Open the drop down list and select the desired stream to use.
Pressures at Which Dew And Bubble are Calculated	Enter up to thirty pressures, at which dew and bubble points are to be calculated. Zeroes entered between pressures will result in linear interpolation of pressures. Pressures must be entered in increasing order. Note: Do not use the interpolation feature if you are entering pressure values using gauge pressures.
Temperature Guess of Bubble Point	Enter a guess for the bubble point temperature at the first pressure specified. Default is feed temperature.
Temperature guess for Of Dew Point	Enter a guess for the dew point temperature at the first pressure specified. Default is feed dew point.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Phase Envelope Examples

There are several sample flowsheets in “Chapter 33: Phase Envelope Samples - c:\designii\samples\equipmnt\phaseenv” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.29: Phase Map

General

The PHAse MAP module calculates dewpoints, bubble points and constant liquid fraction lines for hydrocarbon systems. This module uses improved numerical techniques to perform rigorous calculations in the entire region.

Details

At the present time, the PHAse MAP module will use only APISOAVE, SOAVE, and PENG-ROBINSON equations of state (with or without interaction parameters) for K-value calculations. PHAse MAP calculations will be performed using SOAVEK when any of the following thermodynamic options are specified in the GENeral section: STDK (default), KVAL, RKK, BWRK, and BWRSK. In all cases the critical temperature and pressure are calculated with the Peng-Robinson equation of state. Lines of constant liquid fraction for 0.0 (dew point), 0.50, and 1.0 (bubble point) are automatically calculated. Output will include the critical temperature and pressure of the mixture; temperature, pressure, compressibility factor and enthalpy (in molar units) for each of the bubble and dew points calculated: and a plot of the tabulated temperature pressure points. Temperature, pressure, and compressibility factors will be reported for the other lines of constant liquid fraction.

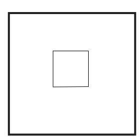
NOTE: The PHAse MAP module does not allow three-phase water hydrocarbon streams.

There are three advantages to using the PHAse MAP module to generate a phase envelope for a hydrocarbon mixture: 1) more accurate prediction of temperatures and pressures for bubble points and dew points including the critical region, and 2) calculation of up to 5 lines of constant liquid fraction within the envelope (3) more efficient calculation for all the data points generated. Moderate amounts of carbon dioxide, nitrogen, and hydrogen sulfide are allowed in the hydrocarbon mixture, but absolutely NO WATER is allowed. Use the Multiple Flashes module to calculate water dew points in a water-hydrocarbon system.

Please see the online **DESIGN II** Help, topic ***Equipment/Phase Map*** or the **DESIGN II** Unit Module Reference ***Guide Chapter 27: Phase Map*** for more details.

Symbols

The Phase Map unit module has one symbol:



The Phase Map module requires that at least one inlet stream and at least one outlet stream be connected to the module.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

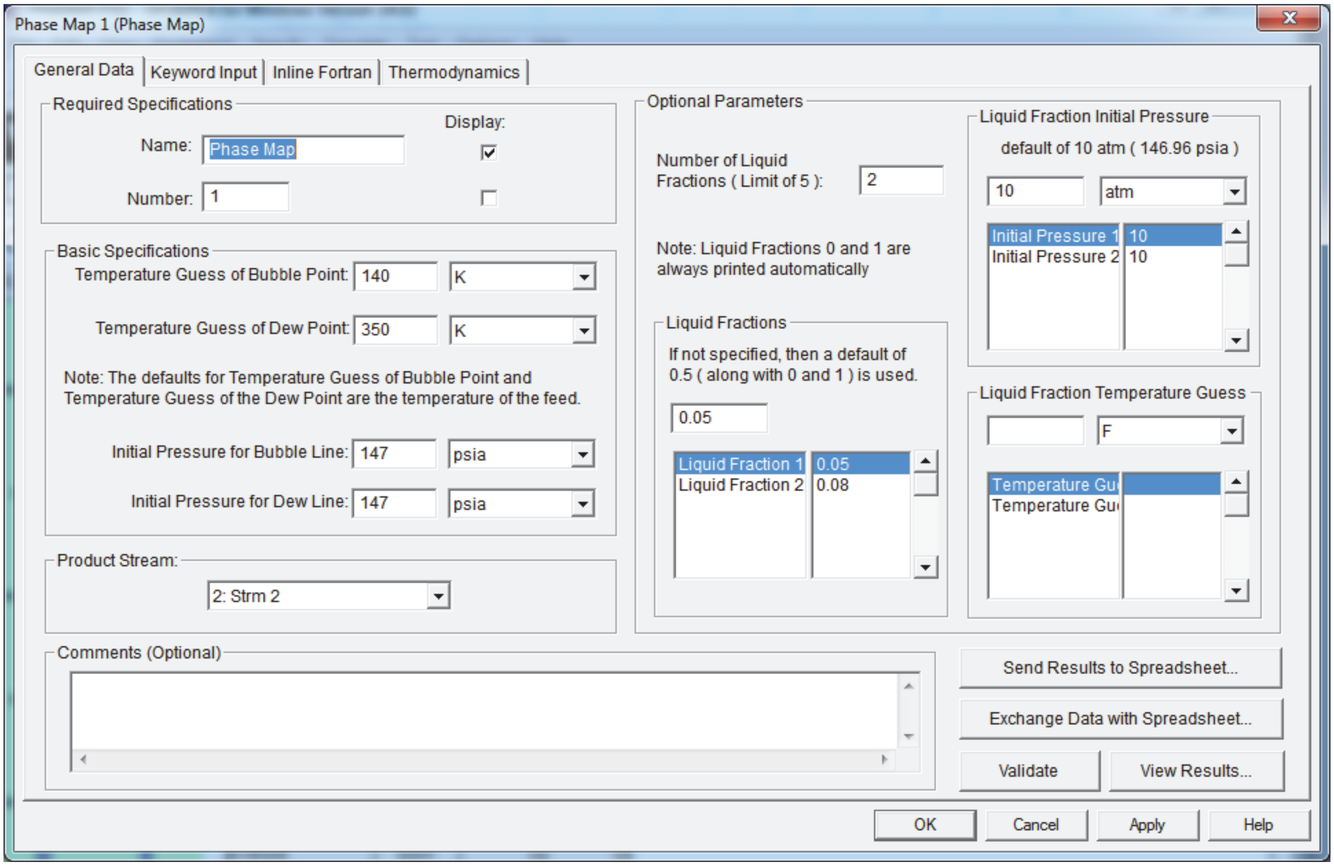


Figure 1: Phase Map Dialog (from phamap.psd)

General Data

Data Items	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Temperature Guess of Bubble Point	Enter guess for the bubble point temperature at 146.96 PSIA (10 atm).
Temperature Guess of Dew Point	Enter guess for the dew point temperature at 146.96 PSIA (10 atm).
Initial Pressure for Bubble Line	Enter the desired starting pressure, for liquid fraction calculations. Use only with Liquid Fraction data. Must have a one-to-one correspondence with values entered in the Liquid Fraction data. Default is 10 atmospheres.
Initial Pressure for Dew Line	Enter the initial pressure for the dew point curve calculations and select a unit.
Number of Liquid Fractions	Enter the number of molar liquid fractions you want calculated, in addition to the default value of 0.50 (you can enter a maximum of five values, including the 0.50 line). Liquid Fractions 0 and 1 are printed automatically.
Liquid Fraction Initial Pressures	Enter the desired starting pressures for liquid fraction calculations. Enter a value and select a unit. To change a value, click on it in the list and type a new value. The default is 10 atm (146.96 psia)
Liquid Fractions	Enter the molar liquid fractions you want calculated in addition to the default value of 0.50 (maximum of 5 values, including the 0.50 line).
Liquid Fraction Temperature Guess	Enter a optional guess for the starting temperature for each liquid fraction specified.

Product Stream Open the drop down list and select the desired stream to use.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Phase Map Examples

There are several sample flowsheets in “Chapter 34: Phase Map Samples - c:\designii\samples\equipmnt\phasemap” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.30: Plate Fin Heat Exchanger

General

The Plate-Fin Exchanger module can be used to simulate multiple stream exchangers. This type of exchangers allows multiple hot and cold streams to have heat exchange with maximum surface area. The module calculates the heat and material balance from known input stream information and the specification to be met.

Details

A typical plate-fin exchanger is composed of a "core" of alternating layers of shaped fins, providing flow channels with a large heat transfer area, and flat separator sheets, which separate these channels from each other.

Incoming streams are split between a number of the flow channels, in a configuration usually determined by the considerations of optimum heat transfer. Each stream may make a number of co or counter-current passes through the core.

The module can be used in either of the two ways:

- without rating, to perform a thermodynamic calculation only. This is the general mode used to calculate the temperature change in the streams and provide enthalpy balances
- with rating, to perform rigorous heat exchange and pressure drop calculations based on the geometry of the exchanger.

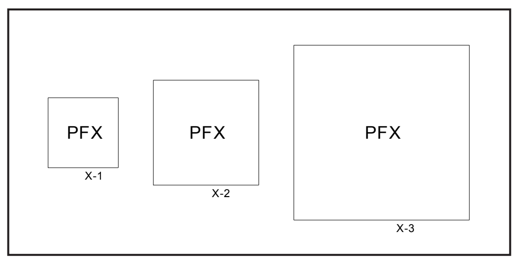
The optional Plate-Fin Exchanger Rating commands allow the rating of the brazed aluminum plate-fin exchangers found in gas processing industry. For example, it can be used to gauge the effect of changing the flow through the exchanger when the exchanger throughput may be a physical constraint on the process optimization.

Please see the online **DESIGN II** Help, topic ***Equipment/Plate Fin Exchanger*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 28: Plate-Fin Exchanger*** for more details.

Symbols

The Plate Fin Heat Exchanger unit module has three symbols, 3x3, 5x5 and 9x9.

You must have at least two inlet streams. You must have at least two outlet streams. You must have the same number of inlet streams and the same number of outlet streams.



Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

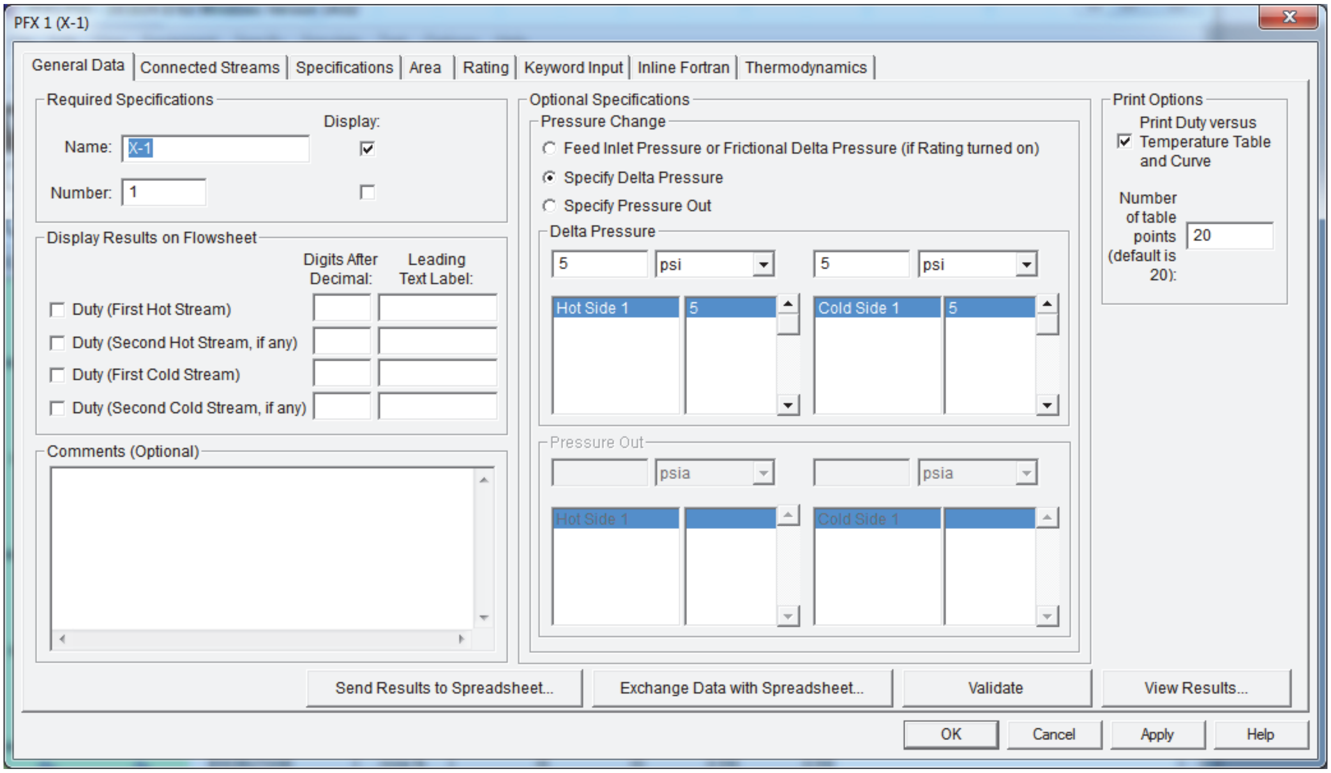


Figure 1: Plate Fin Heat Exchanger Dialog (from pfx1.psd)

General Data

Data Items	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Display Results on Flowsheet	You can optionally display the following results on the flowsheet: Duty (First Hot Stream) Duty (Second Hot Stream, if any) Duty (First Cold Stream) Duty (Second Cold Stream, if any) If you select to display one or more of these results, enter the number of digits to display to the right of the decimal.
Print Options	Select this option to print a Duty versus Temperature Table and Curve on the output. The Q-T table and plot are useful tools for evaluating exchanger design. These curves are based on end-point temperature results. You can quickly determine the minimum temperature approach or locate regions with unrealistically high U x A or temperature crossovers. In addition, bubble points and dew points are marked when vaporization or condensation occurs for either hot or cold streams. Number of table points (default is 20): enter the number of points to use.
Optional Specifications	
Pressure Change	Select one of the following: Feed Inlet Pressure or Frictional Delta Pressure (if Rating is turned on) Specify Delta Pressure Specify Pressure Out If you selected Feed Inlet Pressure or Frictional Delta Pressure: Make sure to select the Rating tab and check the Rate this plate-fin exchanger checkbox and complete the specifications on the tab.

If you selected Specify Delta Pressure:

Select the hot side stream by clicking on its name in the list (if there is more than one stream in the list). Enter the delta pressure change and select a unit. Select the cold side stream by clicking on its name in the list (if there is more than one stream in the list). Enter the delta pressure change and select a unit.

If you selected Specify Pressure Out:

Select the hot side outlet stream by clicking on its name in the list (if there is more than one stream in the list). Enter the pressure out and select a unit. Select the cold side outlet stream by clicking on its name in the list (if there is more than one stream in the list). Enter the pressure out and select a unit.

Connected Streams

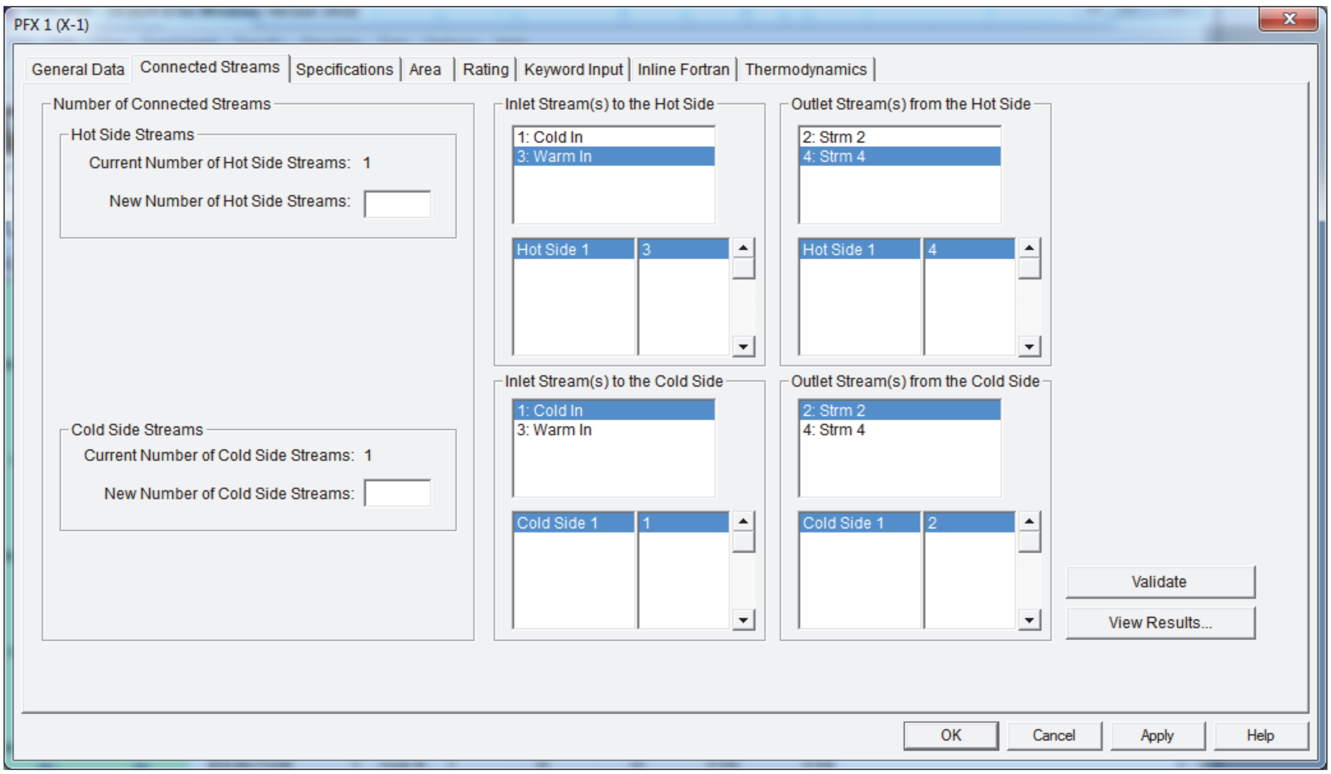


Figure 2: Connected Streams tab (from pfx1.psd)

Data Items	Description
Number of Hot Streams	Enter the number of hot side streams for the plate-fin exchanger. A hot side stream loses heat to other streams.
Number of Cold Streams	Enter the number of cold side streams for the plate-fin exchanger. A cold side stream gains heat from other streams.
Hot Side Inlet & Outlet Stream(s)	Identify the stream numbers in and out of the Plate Fin Exchanger (PFX) connected in the flowsheet for the hot side stream(s).
Cold Side Inlet & Outlet Stream(s)	Identify the stream numbers in and out of the Plate Fin Exchanger (PFX) connected in the flowsheet for the cold side stream(s).

Specifications

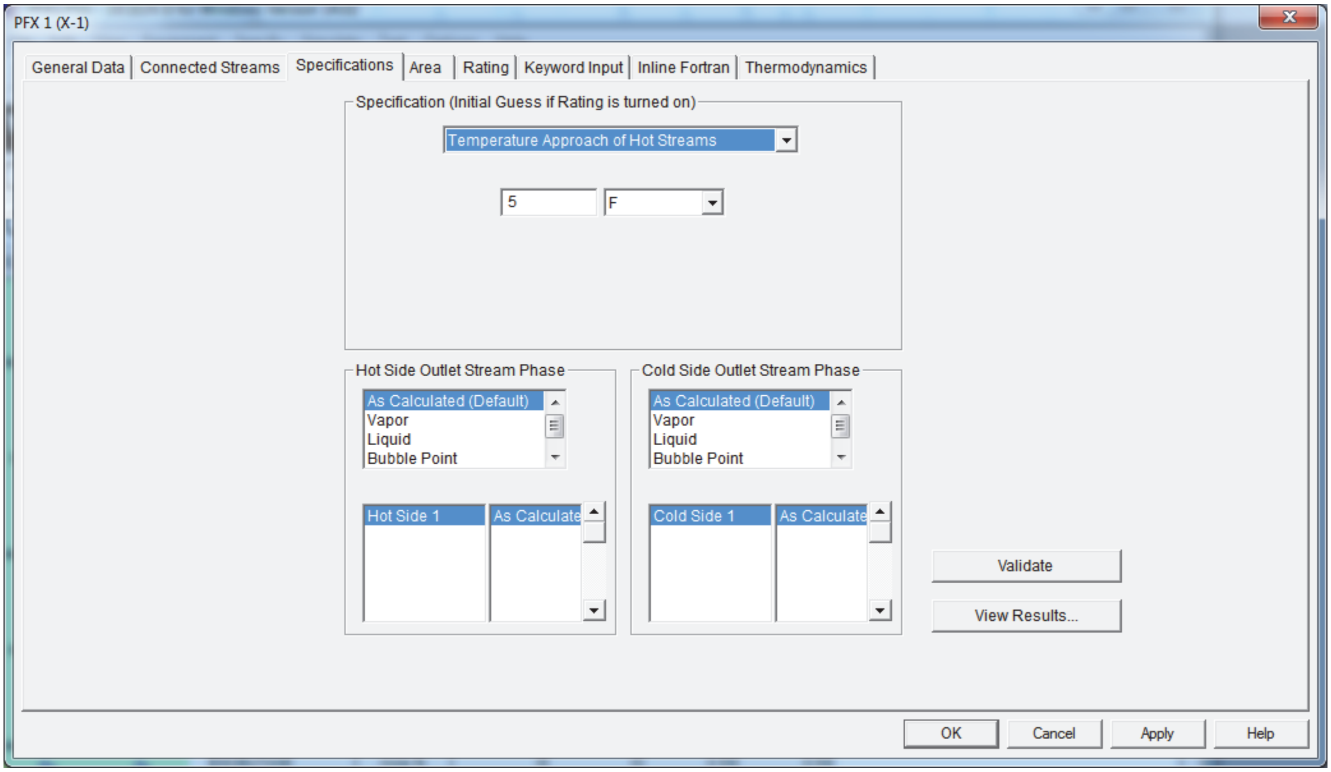


Figure 3: Specification tab (from pfx1.psd)

As Specifications
The following commands are used as specifications when rating is not performed. They will be met as per user specification.

As a Starting Guess
When RATing is done, these commands and data entered will be used as starting initial guesses. Converged temperatures are then obtained for the exit cold and hot side streams based on the plate-fin geometry commands. The default is TEMperature APProach HOT stream = 5.

Data Items	Description
Temperature Out Hot stream	Enter the estimate or specified temperature of the hot (gives up heat) outlet stream.
Temperature Out Cold stream	Enter the estimate or specified temperature of the cold (absorbs heat) outlet stream.
Temperature Approach Hot side	Enter temperature approach desired for cold side outlet stream. Default of 5° F.
Temperature Approach Cold side	Enter temperature approach desired for hot side outlet stream.
Duty of hot side	Enter the desired enthalpy change or duty available for the HOT side stream. This is the duty of each hot side stream of plate-fin exchanger. The overall duty will be the sum of these individual duties.
Outlet Stream Phase	Sets the phase condition of the outlet streams. The phase condition is set for hot side stream outlets first ; then cold side stream outlets. The four choices are from 1 - vapor, 2 - liquid, 3 - bubble point, 4 – dew point. Default is 0 – as calculated. NOTE: The use of this command will override all the above commands.

Area

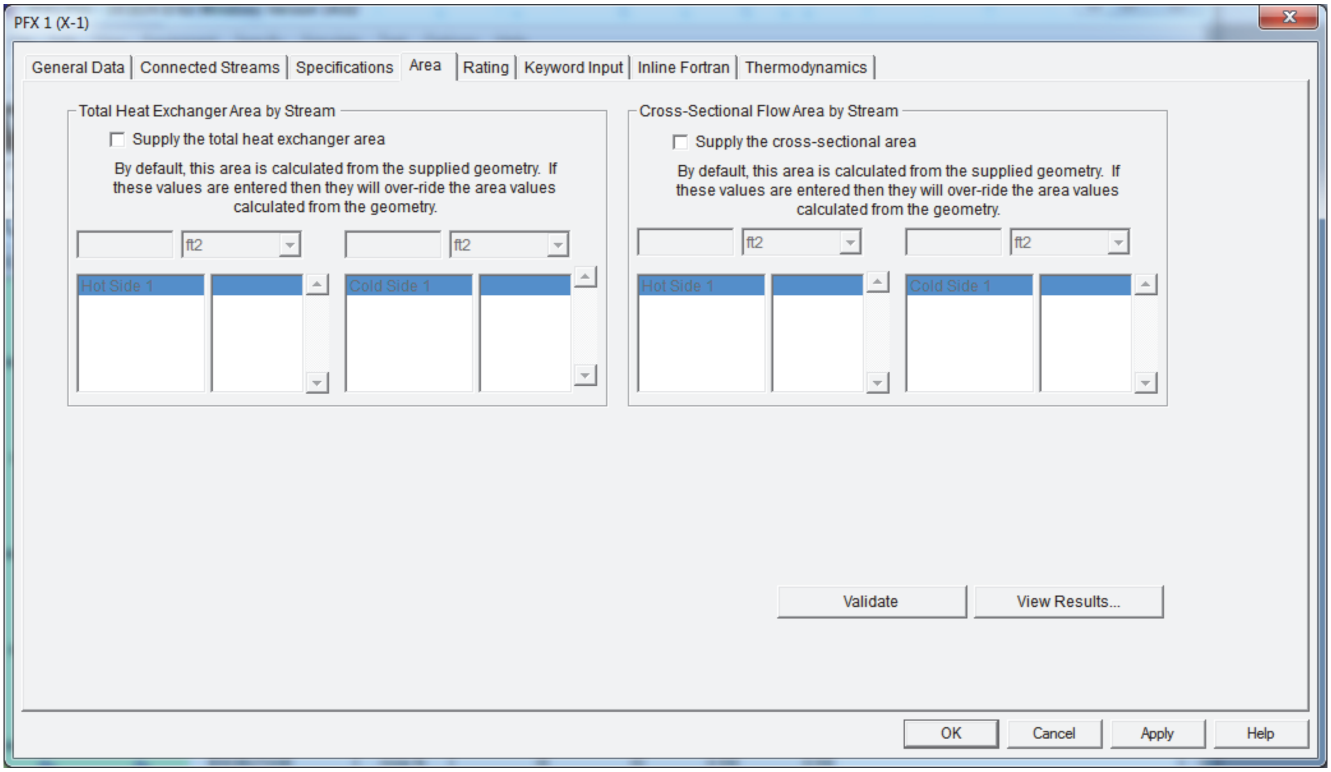


Figure 4: Area tab (from pfx1.psd)

Data Items	Description
Supply the total heat exchanger area	This area is calculated using the supplied geometry. If you select this checkbox and enter values here, they will override the values calculated by the geometry.
Exchange Area	Specify the total heat exchanger area for each stream. By default, the area is calculated from the supplied geometry. If you enter this command the specified value will override the value calculated from the geometry.
Cross-Sectional Flow Area By Stream	Specify the cross-sectional area through which each exchanger stream flows. By default, the area is calculated from the supplied geometry. It is normally not used.

Rating

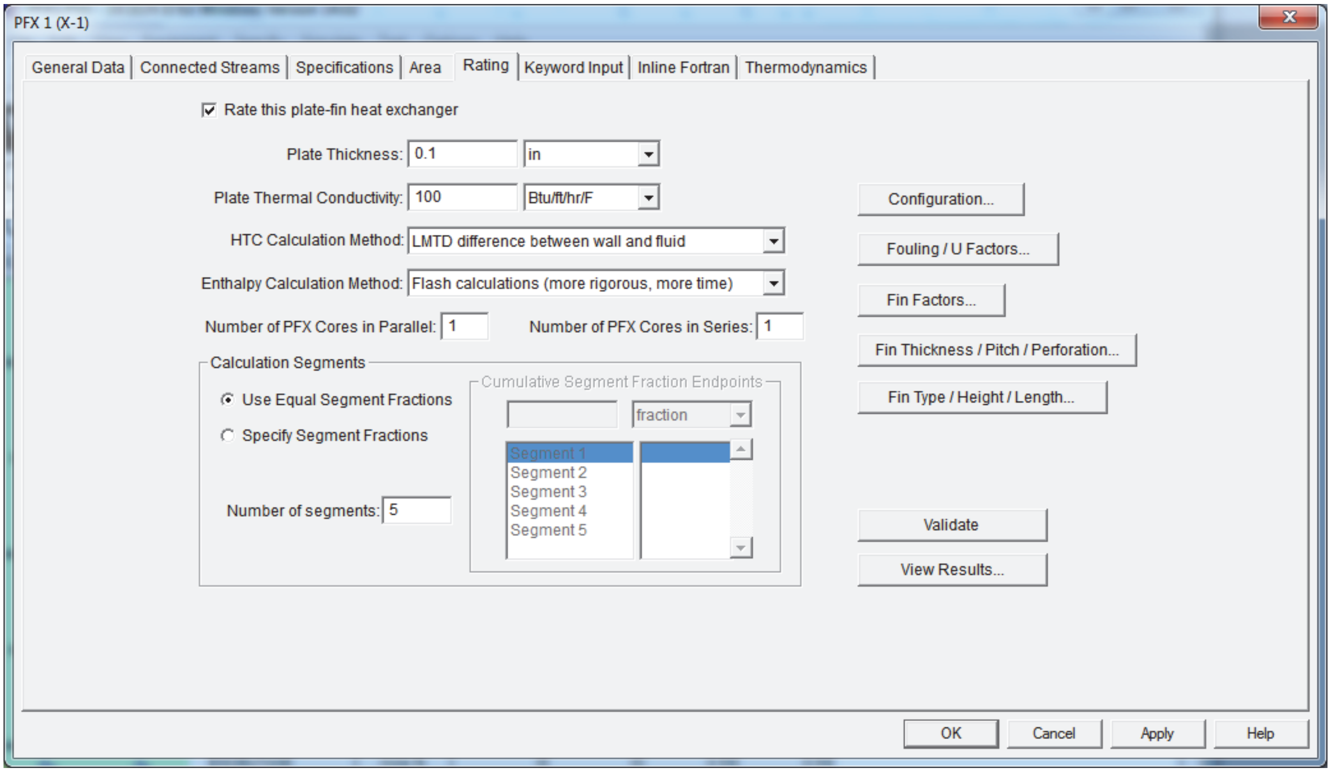


Figure 5: Rating tab (from pfx1.psd)

Data Items	Description
Rating this plate fin heat exchanger	You must check this option ON to perform a rating calculation. NOTE: Turning Rating on ill over-ride the temperature/duty specifications. The outlet stream will be set to the rating results.
Plate Thickness	Specify the parting sheet or plate thickness. The default is 0.08 inches.
Plate Conductivity	Specify the average thermal conductivity of the plate material. The default is 100 BTU/HR/FT/F (English)
Heat Transfer Coefficient Method	This is the calculation method employed for the calculation of overall heat transfer coefficient for each stream. TWO Choices are available: LMTD difference between Wall uses an average cube root mean temperature difference between wall and fluid. Average difference between Wall (default) uses the average difference between the all the fluid temperature. This is normally the most stable method.
Enthalpy Methods	There are two methods to calculate the enthalpy changes occurring in the plate-fin exchanger: Interpolation table to obtain node enthalpies Flash Calculations (more rigorous and more time)
Number of PFX cores in Parallel	Specify the number of plate-fin exchanger cores in parallel. The default is 1. When a value is entered, it is used for all streams.
Number of PFX cores in Series	Specify the number of plate-fin exchanger cores in series. The default is 1. When a value is entered, it is used for all streams.
Configuration	Click this button to set configuration data for the plate fin exchanger.
Fouling/U Factors	Click this button to configure fouling and U Factors.
Fin Factors	Click this button to configure fin factors.
Fin Type/Height/Length	Click this button to configure fin type, height, and length.
Fin Thickness/Pitch/Perforation	Click this button to configure fin thickness, pitch, and perforation.
Calculation Segments	Select either to use equal Segment fractions or to specify segment fractions.

Number in parallel	Specify the number of segments into which the exchanger is to be divided for the rating calculation. The default is 5.
Cumulative Segment Fraction Endpoints	Specify the cumulative length of the increments. The number of values is the number of increments plus one. The default is from zero to one using evenly spaced increments. For instance, if NUM INC = 5 then the default is "INC FRA = 0, 0.2, 0.4, 0.6, 0.8,1".

Configuration

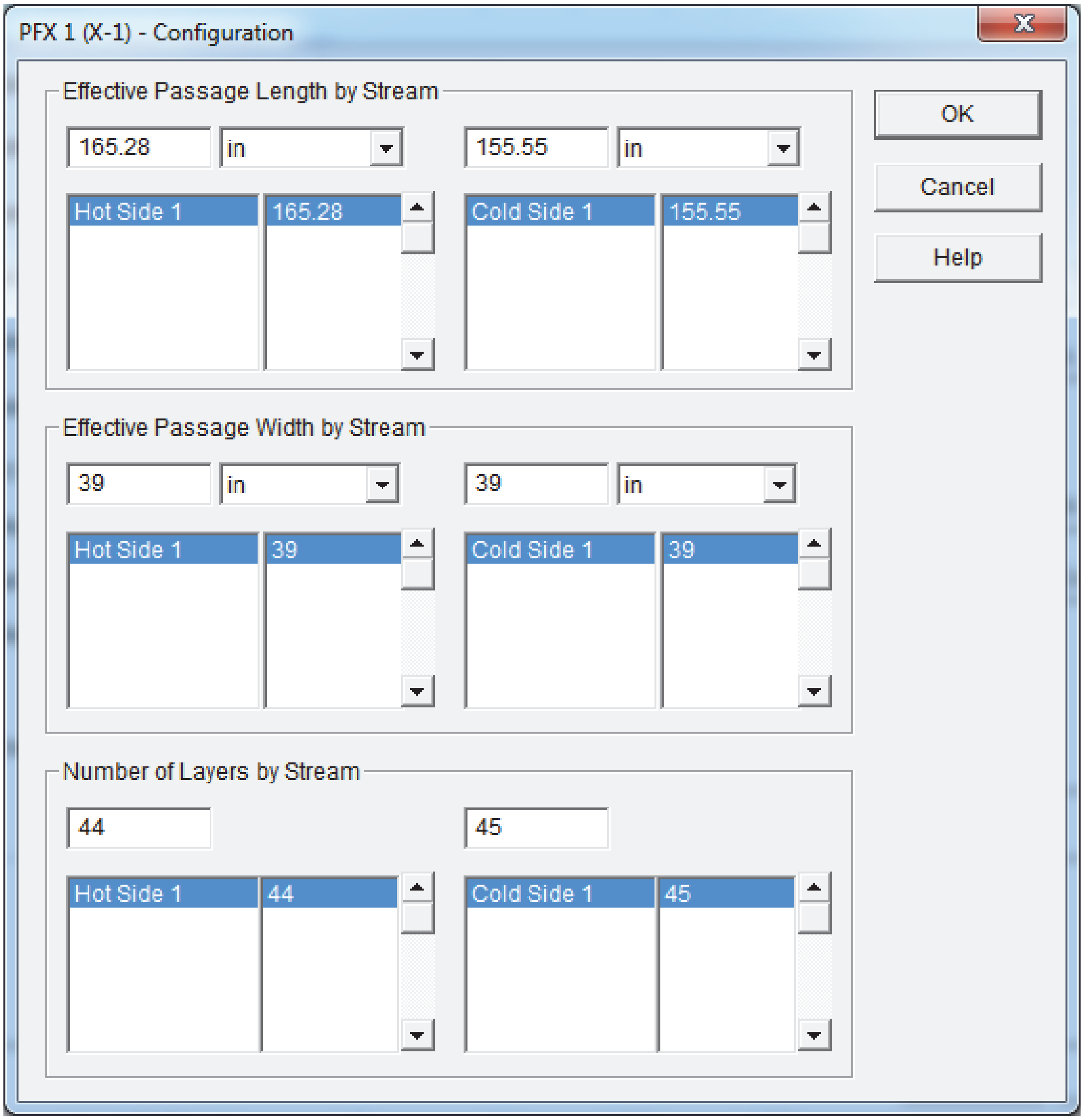


Figure 6: Configuration Dialog (from pfx1.psd)

The configuration commands are used to describe the physical configuration of the Plate-Fin Exchanger (PFX).

Data Items	Description
Effective Passage Length	Specify the effective passage length along the direction of flow that each stream is in the effective heat exchange zone. Default is 120 inches (10 ft,) implying an exchanger 10 ft long.
Effective Passage Width	Specify the effective passage width perpendicular to the direction Of flow that each stream is in the effective heat exchange zone. Default is 36 inches (3 ft), implying an exchanger 3 ft long.
Number of Layers	Specify the number of finned layers or flow passages per stream. The default value for each stream is 20, meaning that each stream flows through 20 inter-plate spaces for each pass through the exchanger.

Fouling/U Factors

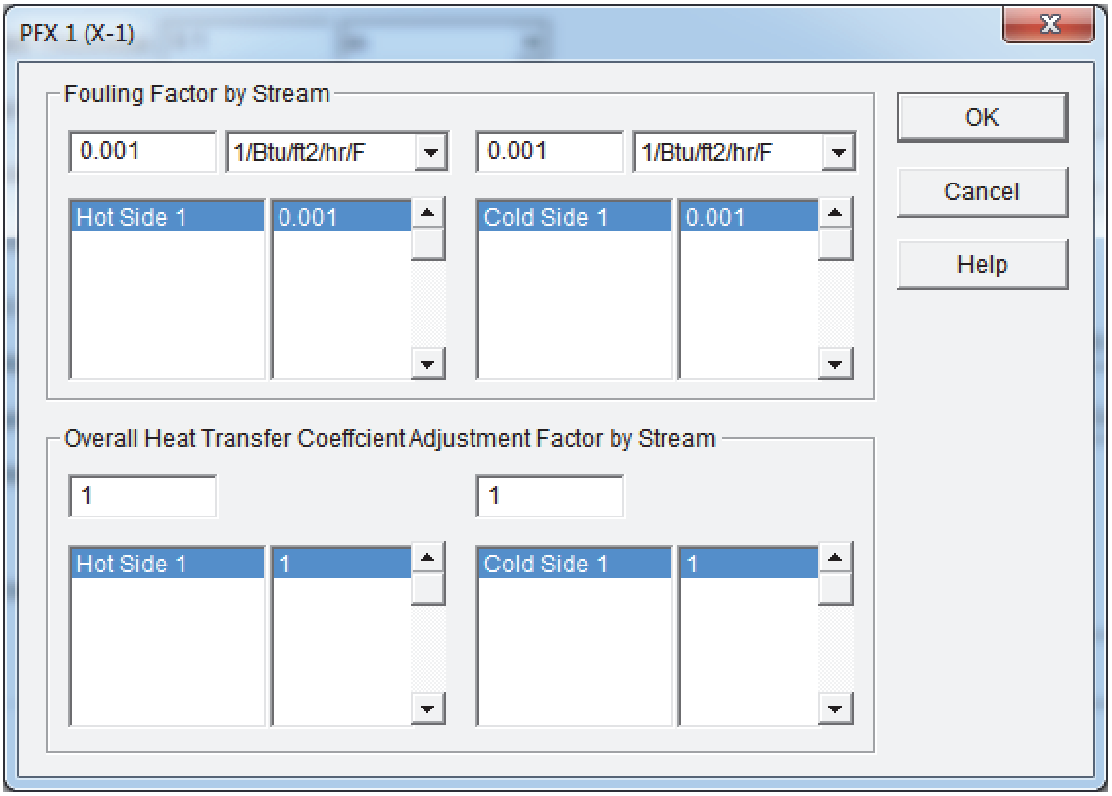


Figure 7: Fouling/U Factors Dialog (from pfx1.psd)

Data Items	Description
Fouling Factor	Enter the fouling factor for each stream. Default is 0.001 hr-ft2-/btu.
Overall Heat Transfer Coefficient Adjustment Factor	Specify a factor to adjust the Overall Heat Transfer Coefficient. It provides a "de-rating" factor for each stream which will be multiplied with the calculated heat transfer coefficient. The default is 1.

Fin Factors

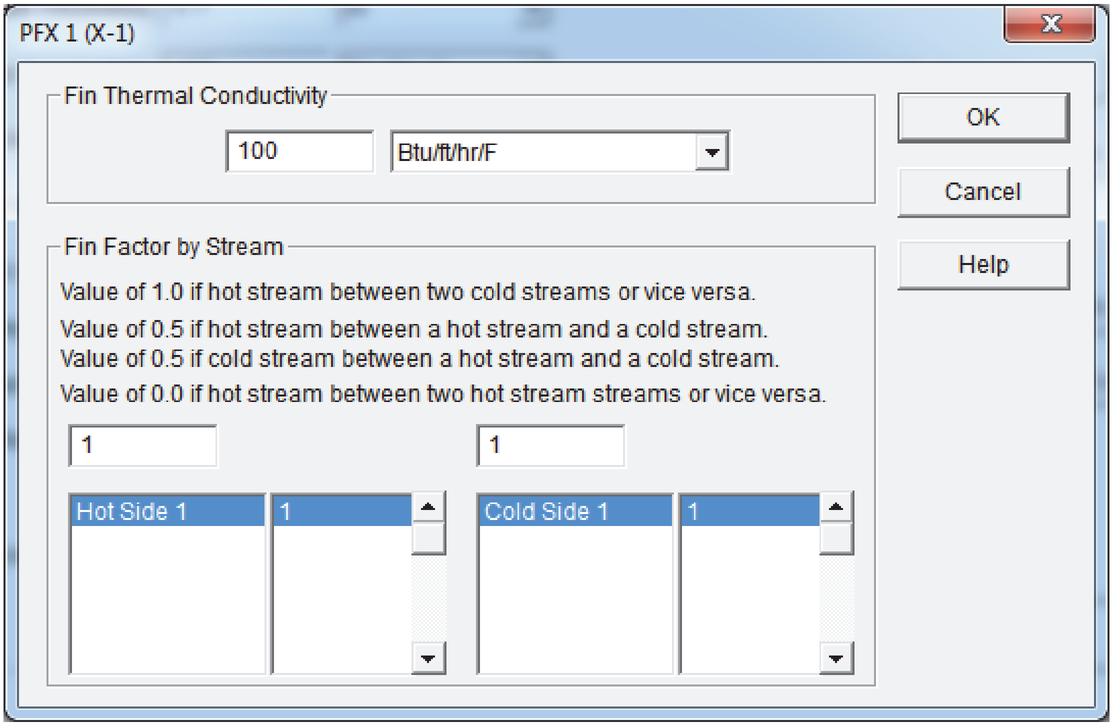


Figure 8: Fin Factor / Conductivity (from pfx1.psd)

Data Items	Description
Fin Thermal Conductivity	Specify the average thermal conductivity of the fin material. Default is 100 BTU/HR/FT/F (English)
Fin Factor	Specify a fractional factor for each stream which characterizes the fin efficiency (measure of effectiveness of the fin for heat transfer to the fluid channel) based on the exchanger stacking arrangement. Default is 1.

Fin Type/Height/Length

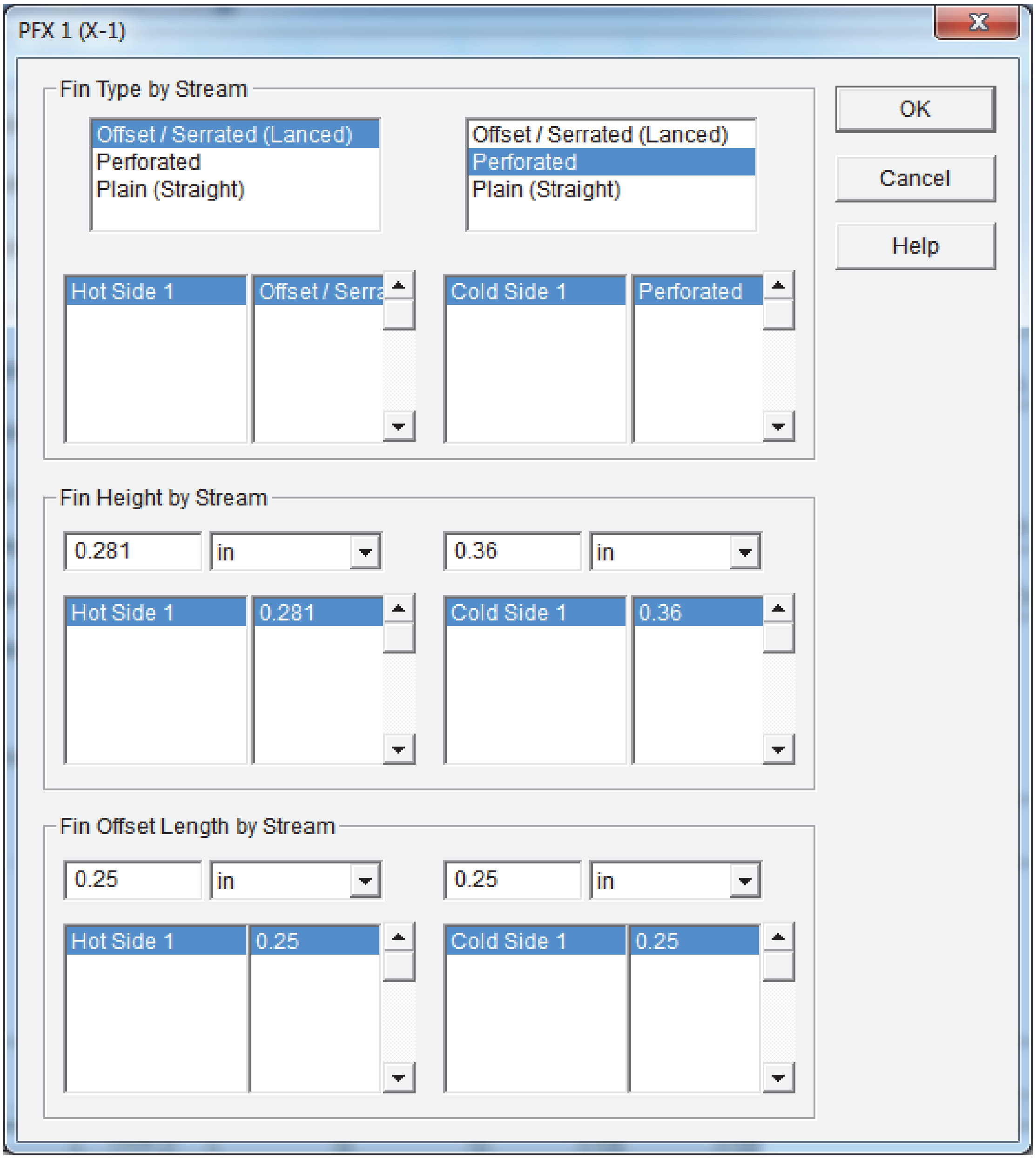


Figure 9: Fin Type / Height / Offset Length (from pfx1.psd)

Data Items	Description
Fin Type	Specify the fin type of the plate-fin exchanger. Choices available are: 1 - offset/serrated (lanced); 2 - Perforated, 3 - Plain (straight). Default is 1
Fin Height	Specify the fin height. The default is 0.28 inches
Fin Offset Length	Specify the fin length. This command is applicable when offset fins are used for stream passages. The default is 0.25 inches.

Fin Thickness/Pitch/Perforation

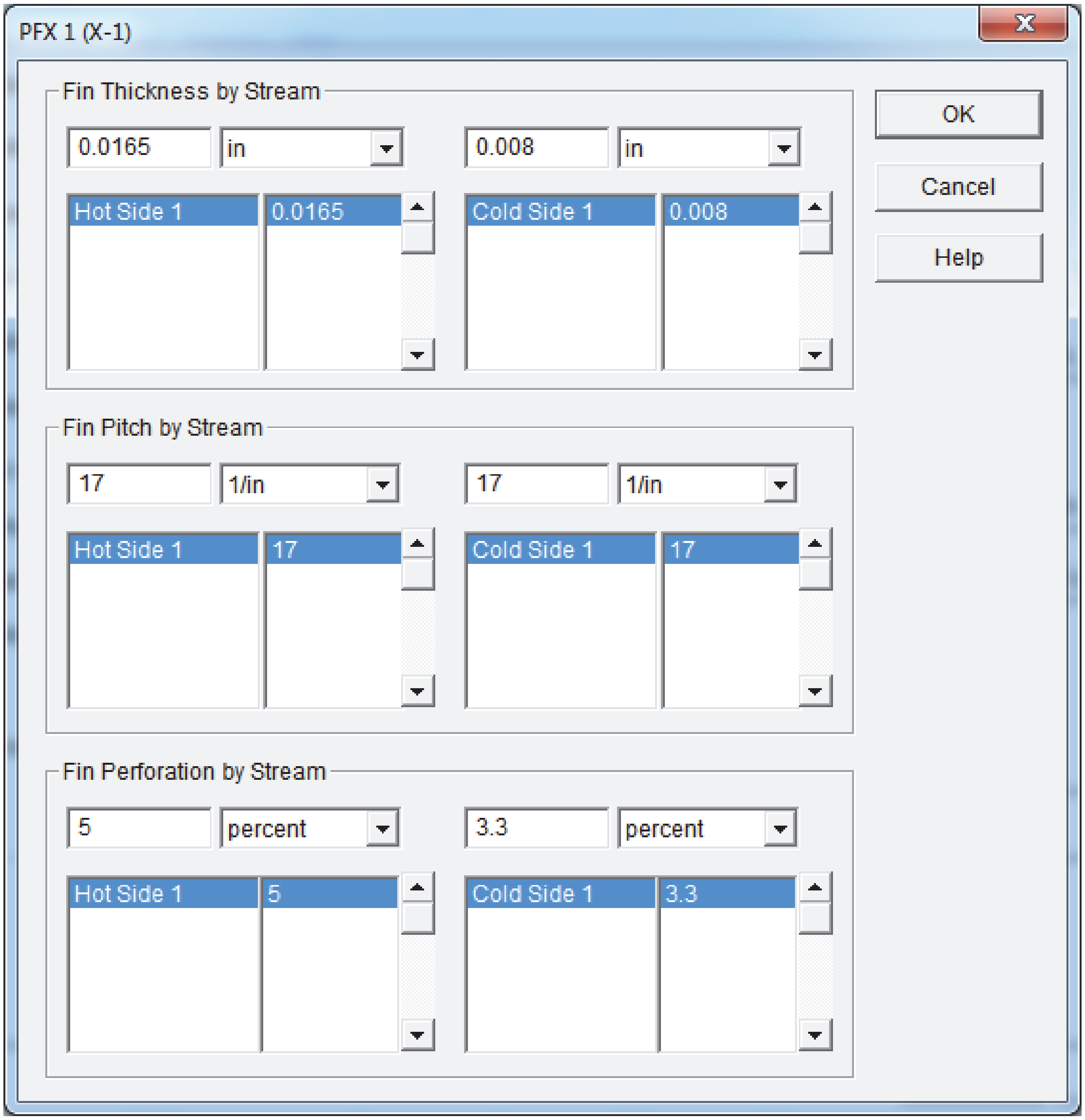


Figure 10: Fin Thickness / Pitch / Perforation (from pfx1.psd)

Data Items	Description
Fin Thickness	Specify the thickness of fin metal. The default is 0.0013 ft (0.016 in). Typical heat transfer fin thickness range from 0.006 to 0.023 inches.
Fin Pitch	Specify the fin spacing or number of fins per inch measured perpendicular to the direction of flow of each stream. The default is 17 fins per inch (204 per ft).
Fin Perforation	Specify the fin perforation percent. The default is 5 percent.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Plate-Fin Exchanger Examples

There are several sample flowsheets in “Chapter 34: Phase Map Samples - c:\designii\samples\equipmnt\phasemap” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.31: Plug Flow Reactor

General

The Plug Flow Reactor module calculates single-phase reactions using specified information on the reactor dimensions, stoichiometric coefficients of the reactants and products, and power law rate expressions. Alternate methods for specifying reaction kinetics are available in the form of keyword commands or using Inline Fortran. Heat of formation data is required for any component with an ID number greater than 98. Enter this data in the GENeral section of the input file via the HEAt of FORmation command.

When the plug flow reactor module is used, only a single stream for the reactor feed may be coded. If a cooling stream is required, it must be coded as the second inlet stream. Limiting assumptions by the module include no radial mixing, concentrations of reacting species vary only along the length of the reactor, and except for isothermal operation, the temperature varies along the length of the reactor. Default values will be used where indicated if no value is entered by the user.

Details

The four types of PFR that can be modeled are: Isothermal, Temperature profile, Adiabatic or Cocurrent coolant.

Results
PFR results will be reported in the "NOW CALLING" section of the output file (mass basis) and compositions for the product stream will be in the Stream Summary and Detailed Stream Report. Reactions and overall duty for the reactor will be reported in the Equipment Summary.

Method of Calculation
The default rate expressions are the power law type with the rate constant expressed by the Arrhenius equation:

K = Ko * E^(-Eo / RT)

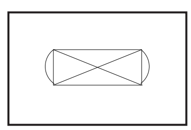
where Ko is the frequency factor, Eo is the activation energy, R is the molar gas constant and T is the temperature. Alternatively, rate expressions can be entered with Inline Fortran statements using the DURing option. The reactions are integrated along the length of the reactor using the Gear integration method.

Please see the online **DESIGN II** Help, topic ***Equipment/Plug Flow Reactor*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 29: Plug Flow Reactor*** for more details.

Symbols

The Plug Flow Reactor unit module has one symbol:

The Plug Flow Reactor module requires that at least one inlet stream and at least one outlet stream be connected to the module.



Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

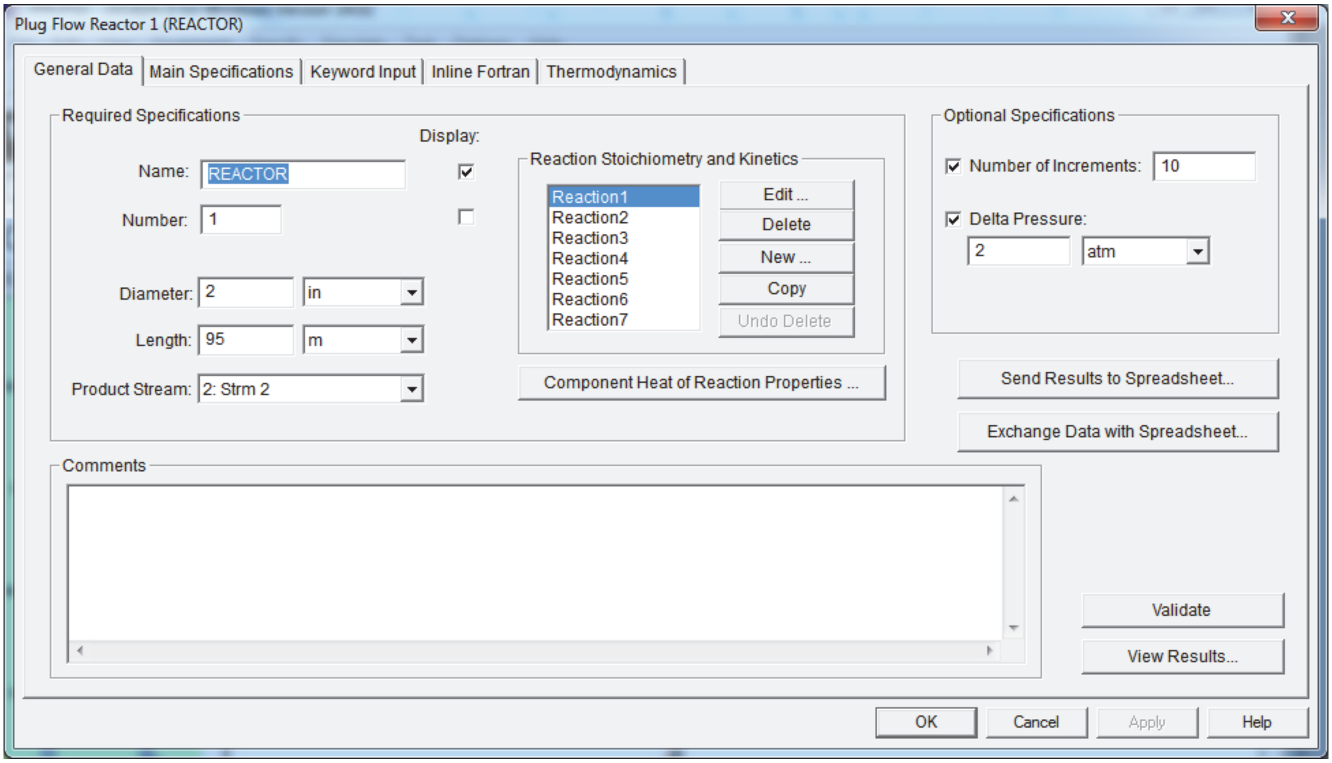


Figure 1: General Data for Plug Flow Reactor (from pfr.psd)

Data Items	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Diameter/Length	Enter the internal diameter and length of the reactor and select a unit.
Product Stream	Open the drop down list and select the desired stream
Reaction Stoichiometry and Kinetics	Enter the reaction to use. Click the New button to display the Reaction dialog that you can use to create new reaction stoichiometry and kinetics specifications.
Component Heat of Reaction Properties	Click this button to display the Thermodynamic and Transport Methods dialog with the Component Heat of Reaction Properties tab displayed.
Number of increments	Select this optional specification to show Compositions, temperature, and pressure for the reactor inlet, outlet, and several intermediate points. The default value is 10.
Delta pressure	Enter a pressure drop for the reactor and select a unit.

.

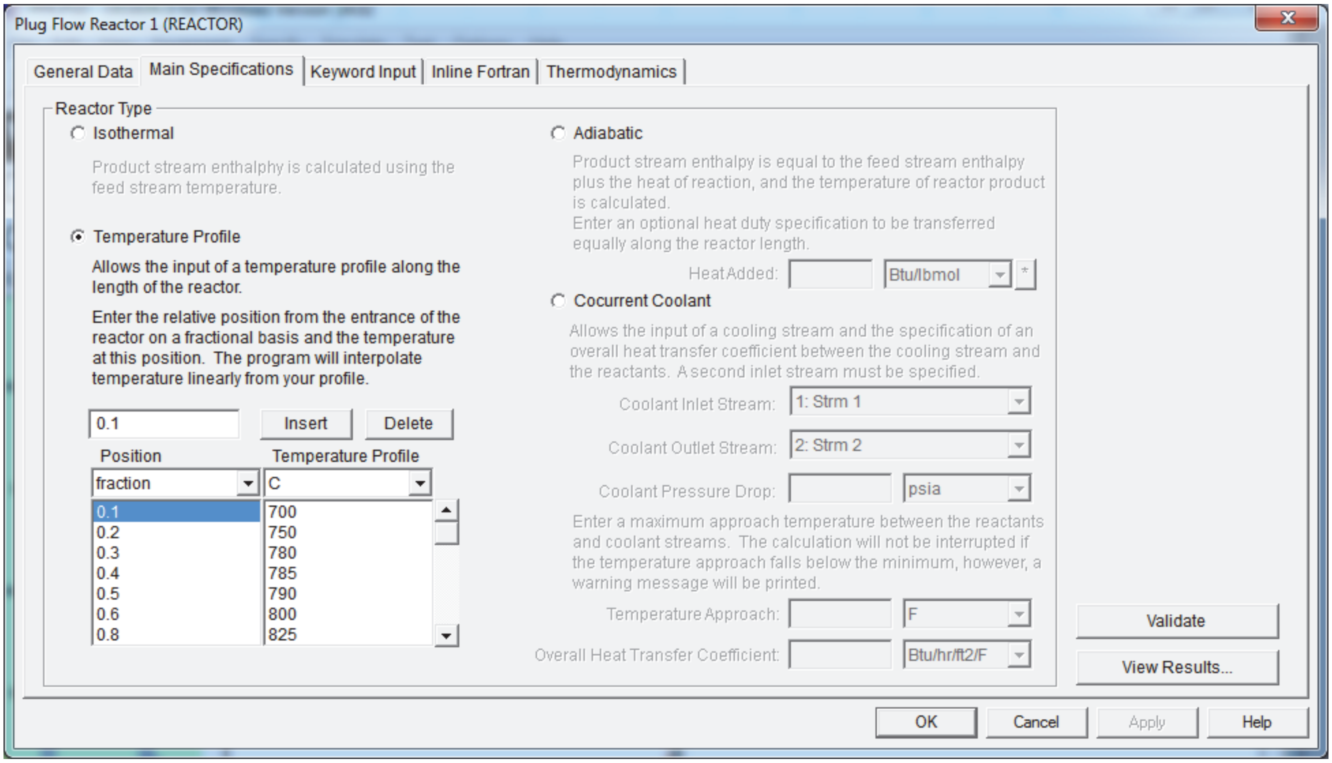


Figure 2: Main Specifications for Plug Flow Reactor (from pfr.psd)

This dialog is organized into four radio button selection items that tells you how many PFR specifications types you are allowed. When you attempt to select one, the other boxes will be turned off.

Data Item	Description
Isothermal	No additional commands are required for this reactor.
Temperature Profile	Enter the temperature profile where “Position" is the relative position from the entrance of the reactor on a fractional basis and "Temperature Profile" is the temperature at position "Position".
Adiabatic	Enter an optional heat duty specification to be transferred equally along the reactor length.
Cocurrent Coolant	Enter a stream number to be used as a coolant stream. Enter a pressure drop for the coolant stream. Enter a minimum approach temperature between the reactants and coolant streams. Enter the overall heat transfer coefficient between the coolant stream and the reactor.

Reaction

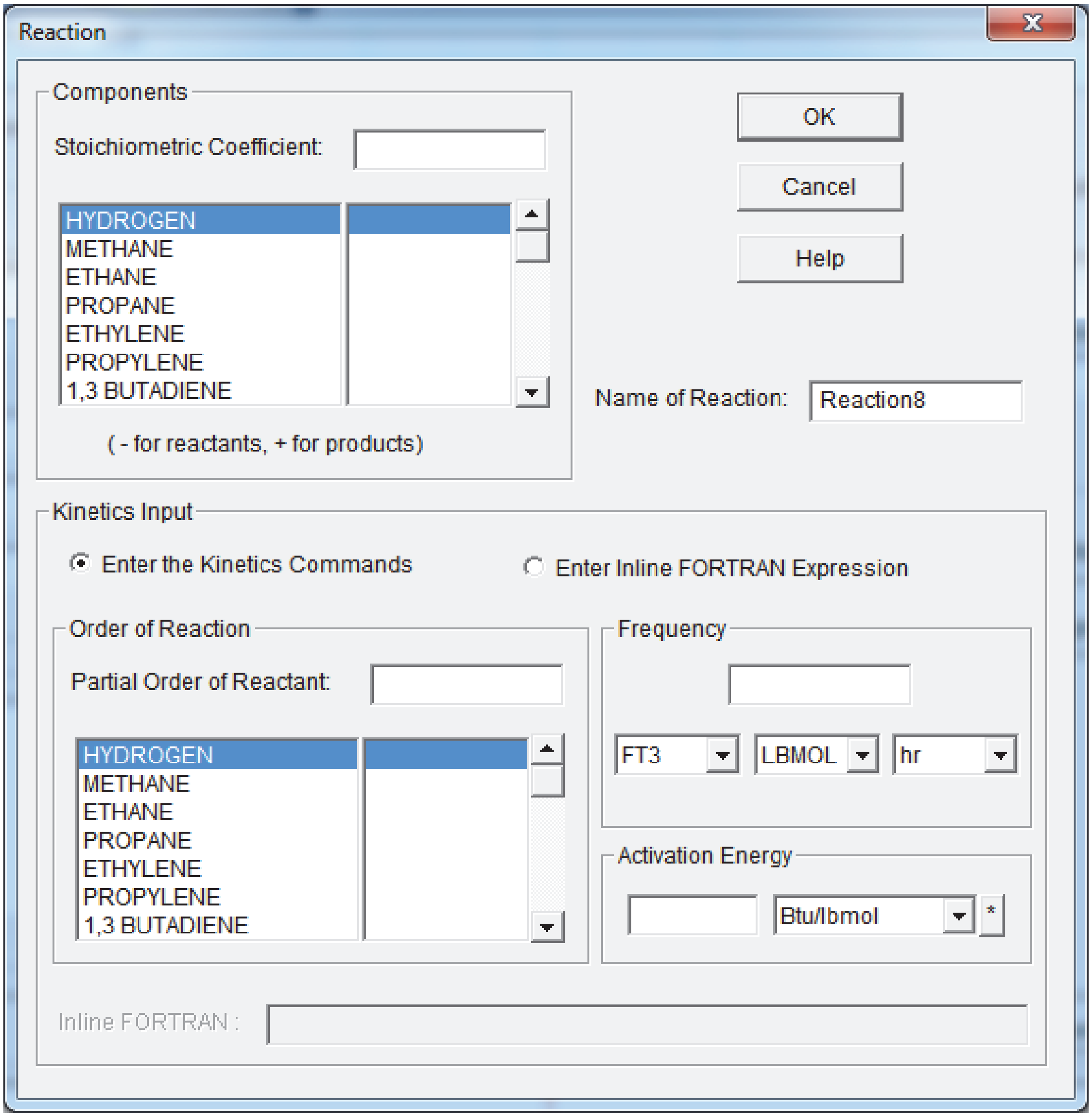


Figure 3: Reaction dialog for Plug Flow Reactor (from pfr.psd)

Data Item	Description
Stoichiometric Coefficient	Select a component from the list. Enter a negative coefficient number to indicate a reactant or a positive coefficient number to indicate a product. If you do not enter a value, the component is treated as inert.
Name of Reaction	View the name to use for the reaction, or you can enter a new one.
Kinetics Input	Choose either Enter the Kinetics Commands or Enter Inline FORTRAN Expressions.
Enter the Kinetics Commands	
Order of Reaction	Select a component from the list. Enter the partial order of the reactant.
Frequency	Enter the frequency factor. Open the drop down list and select the desired units.
Activation Energy	Enter the activation energy and select a unit.
Enter Inline FORTRAN Expressions	
Inline FORTRAN	You can enter FORTRAN to define the reaction kinetics.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. In future releases of the product, these instructions will be replaced with fill-in-the-blank dialog boxes. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Plug Flow Reactor Examples

There are several sample flowsheets in “Chapter 36: Plug Flow Reactor Samples - c:\designii\samples\equipmnt\plugreac” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.32: Polytropic Compressor

General

The Polytropic Compressor module is used to simulate a single stage of a centrifugal compressor based on manufacturer's performance curves. When it is used in conjunction with other DESIGN II modules, it is possible to simulate multistage compression with intercooling, recycle or kickback between stages, and to evaluate existing compressors under off-design conditions. For multistage compression, a separate POLytropic compressor module is used to simulate each stage. Each stage can have several wheels. HEAt EXChanger, FLAsh, and other modules can be added between stages for intercooling and liquid dropout effects.

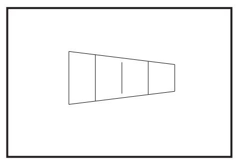
Details

The polytropic efficiency and polytropic head (work/mass/rotational speed) must be provided as a function of capacity (inlet volumetric flowrate/rotational speed). If manufacturer's data is in a different form, the data can be recalculated and plotted into the required form. The POLy module calculates the outlet temperature and pressure, and the work consumed for a given suction stream and desired impeller speed. The speed can also be varied by the CONTroller module to achieve a desired outlet condition (e.g. pressure or temperature). Interpolation at the actual speed point will be performed when a specific speed point is entered between multiple speed points.

Please see the online **DESIGN II** Help, topic ***Equipment/Polytropic Compressor*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 30: Polytropic Compressor*** for more details.

Symbols

The Polytropic Compressor unit module has one symbol:



The Polytropic Compressor module requires that at least one inlet stream and at least one outlet stream be connected to the module.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

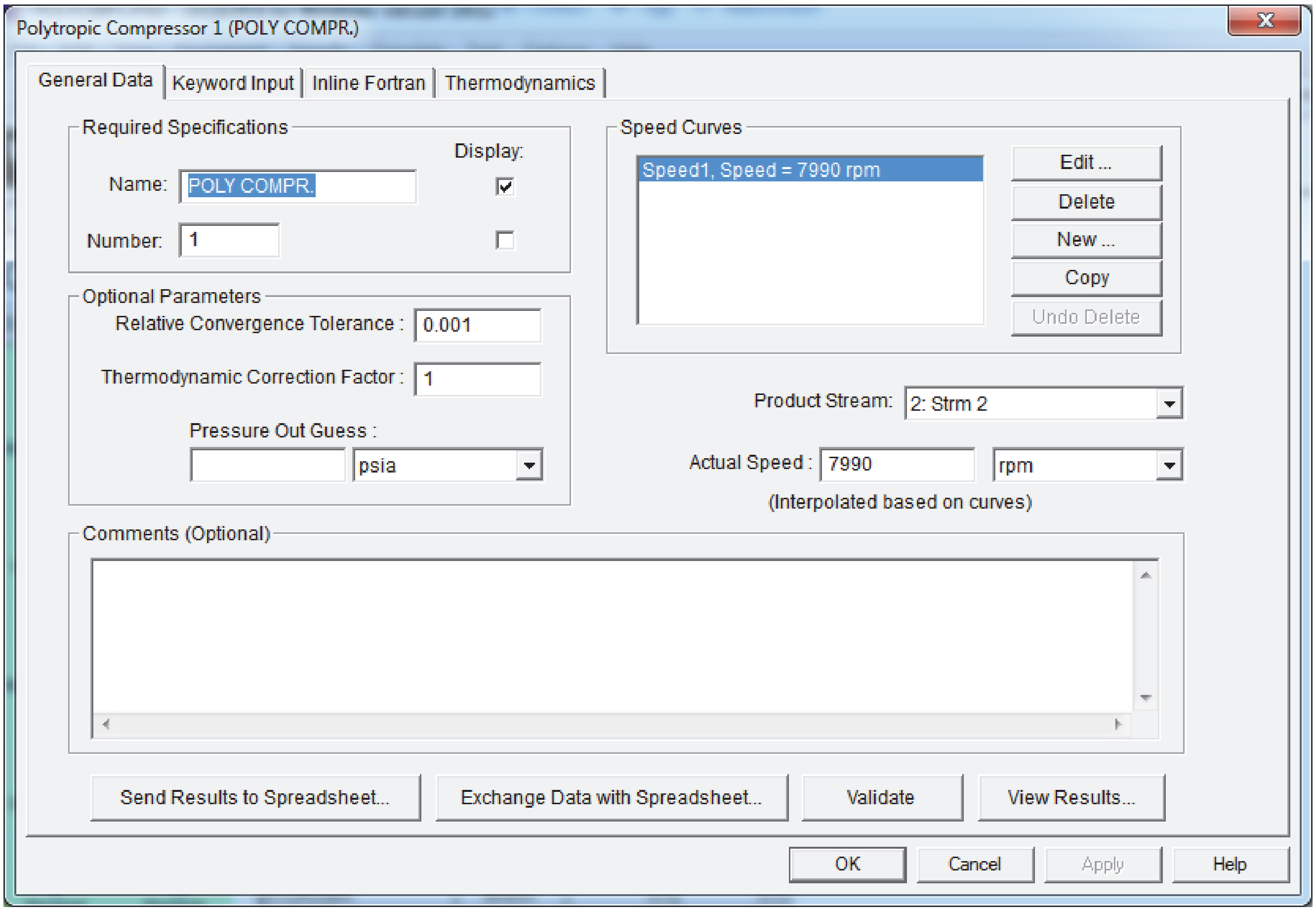


Figure 1: Polytropic Compressor Dialog (from polycmp1.psd)

General Data

Data Items	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Speed Curves	You can create speed curves for the compressor. Click the New button to display the Speed Curves dialog that you can use to create a new speed curve. Select an existing curve and click the Edit button to modify the curve's parameters. Select an existing curve and click the Delete button to remove that curve. Select an existing curve and click the Copy button to replicate the curve. You can then use the Edit function to make any minor changes to the copied curve.
Optional Parameters Relative Convergence Tolerance	Enter the relative tolerance for convergence.
Thermodynamic Correction Factor	Enter a factor (usually between 0.9 and 1.1) to correct the manufacturer's curves for the thermodynamic option you select.
Pressure Out Guess	Enter a guess for the discharge pressure and select a unit. The default is 2.5 times the inlet pressure.
Product Stream	Open the drop down list and select the desired stream.
Actual Speed	The actual speed is interpolated based on the curves. You can change the value and select a unit.

Speed Curves

This dialog is used to enter the speed curve specifications for the polytropic compressor. For multiple speed curves, interpolation at the actual speed point will be performed when a specific speed point is entered between multiple speed points.

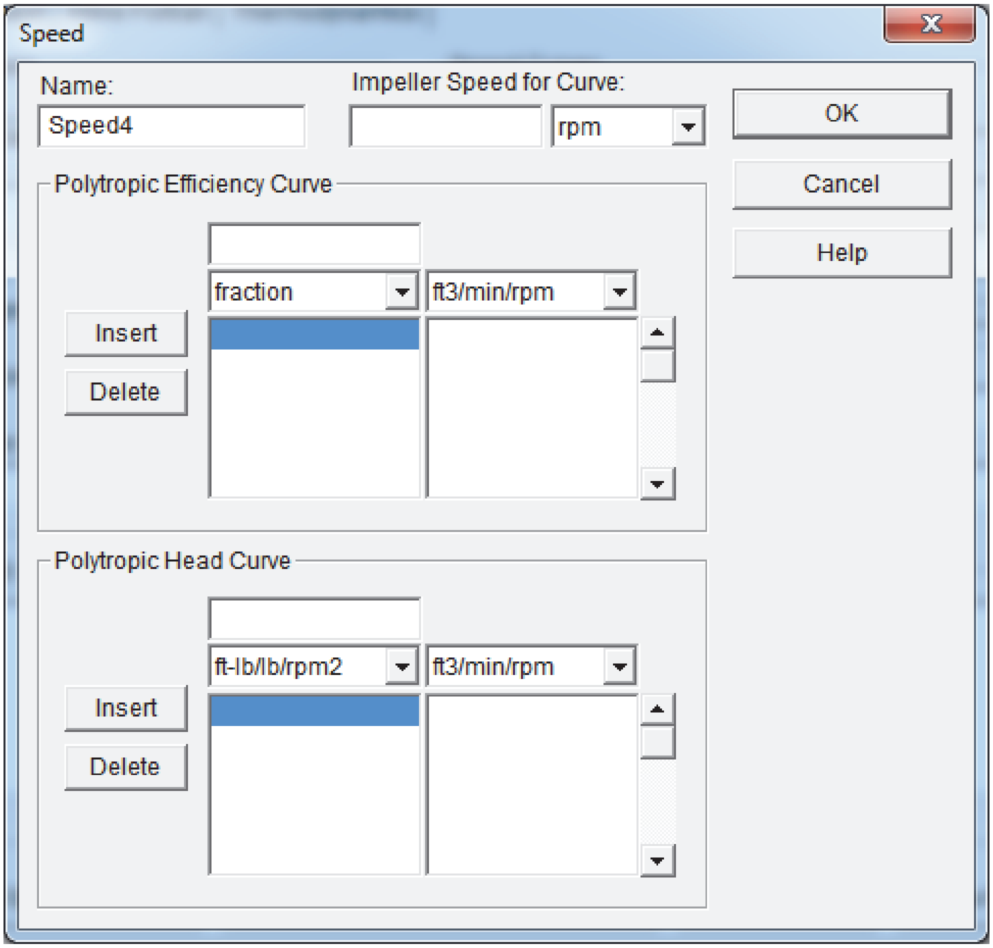


Figure 21: Polytropic Compressor Dialog (from polycmp1.psd)

Data Items	Description
Name	The name of the speed curve. You can type a new name.
Impeller Speed for Curve	Enter the rotational velocity for the impeller and select a unit.
Polytropic Efficiency Curve	Enter a polytropic efficiency versus capacity (actual volume flowrate/speed) point and select the desired units. Click the Insert button to add the point. You can enter up to 30 points. To remove a point, click on its name in the list then click the Delete button.
Polytropic Head Curve	Enter a polytropic head versus capacity point and select the desired units. Click the Insert button to add the point. You can enter up to 30 points. To remove a point, click on its name in the list then click the Delete button.

Required Specifications: Keyword Input

Click the Keyword Input tab. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Polytropic Compressor Examples

There are several sample flowsheets in “Chapter 37: Polytropic Compressor Samples - c:\designii\samples\equipmnt\polycomp” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.33: Pump

General

The Pump is used to pump a liquid to a specified outlet pressure or to a pressure limited by a specified work available.

Details

The Pump module pumps a liquid to a specified outlet pressure or to a pressure limited by a specified work available. Centrifugal pumps are modeled by calculating the work from the enthalpy for isentropic compression and dividing by the efficiency. Reciprocating pumps are normally modeled by specifying a volumetric efficiency. The volumetric efficiency you enter does not take into account the mechanical efficiency.

Only one inlet and one outlet stream may be connected to the Pump. You can use the Mixer module to mix streams.

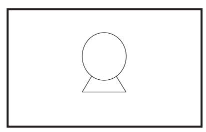
When vapor is found in the suction of the pump, a warning message is printed in the DESIGN II output. If the entropy calculation is specified, a rigorous mixed-phase calculation is performed. If the entropy calculation is not specified, the vapor is pumped as if it were a liquid. If the vapor fraction of the feed is greater than 0.9, the calculation switches to the compressor option and a significant rise in calculated temperature may occur.

Please see the online **DESIGN II** Help, topic ***Equipment/Pump*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 31: Pump*** for more details.

Symbols

The Pump unit module has one symbol:

The Pump module requires that one inlet and one outlet stream be connected to the module.



Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

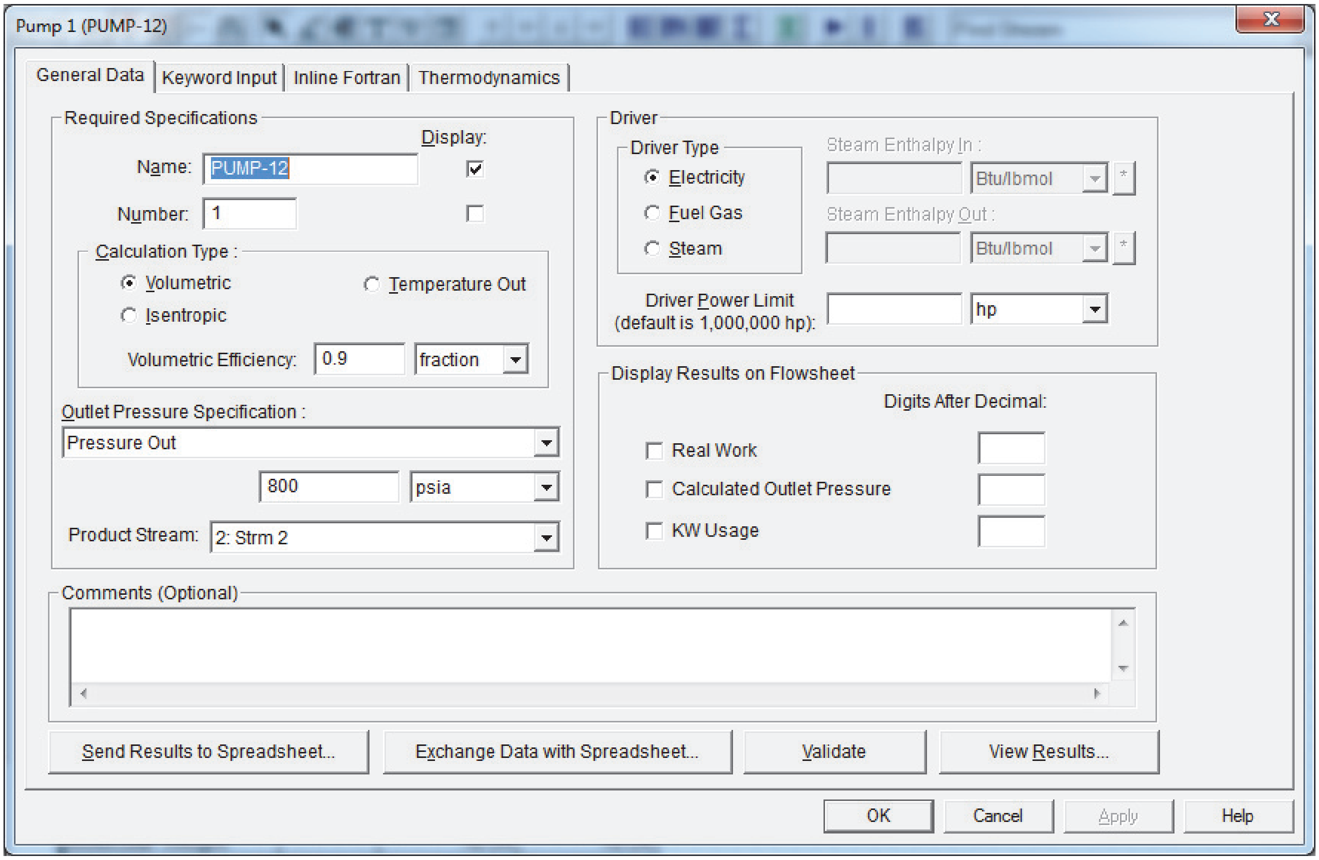


Figure 1: Pump Dialog (from pump1.psd)

General Data

Calculation Type & Outlet Pressure Specification

This dialog is used to enter the basic specifications for the Pump Module.

Data Items	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Calculation Type	
Volumetric	This is normally used for a reciprocating pump. With this specification, the efficiency is the volumetric efficiency. It will not account for the mechanical efficiency. For example, if you are modeling a 25 HP pump with a 90% mechanical efficiency; then the work available for the pump is .9(25)= 22.5 HP.
Isentropic	This is normally used for a centrifugal pump. With this specification, the enthalpy change on the inlet stream resulting from isentropic compression will be divided by the efficiency to calculate the work.
Volumetric Efficiency	The pump efficiency. This specification has the meaning stated above for the Volumetric and Isentropic specifications.
Outlet Pressure Specification	
Pressure Out	The discharge pressure of the pump.
Delta Pressure	The desired pressure change or delta pressure. The default is twice the suction pressure.
Dew Point Temperature	The dew point pressure at the specified temperature is calculated ; then used as the discharge pressure for the pump calculation.
Bubble Point Temperature	The bubble point pressure at the specified temperature is calculated ; then used as the discharge pressure for the pump calculation.

Product Stream	Open the drop down list and select the desired stream.
Driver	Select this dialog for driver specifications.
Fuel Gas	Specifies that the driver for the pump is based on fuel gas. The amount of fuel gas consumed assumes 980 BTU/SCF (8.639 E+05 KJ/KGMOL) and 8000 BTU/HR (3.14 E-03 KJ/SEC/WA).
Electricity	Specifies that the pump is driven electrically. DESIGN II will report the kilowatts required for the electric motor.
Steam	Specifies that the pump is stream driven. If you choose this driver, you need to enter the inlet and outlet enthalpy for the steam.
Steam Enthalpy In	The enthalpy of the steam at the inlet of the driver.
Steam Enthalpy Out	The enthalpy of the steam at the outlet of the driver.
Driver Power Limit	The available work capacity. If the available work is insufficient to meet the specified outlet pressure, the maximum pressure that can be achieved is calculated.
Display Results on Flowsheet	You can optionally display the following results on the flowsheet: Real Work Calculated Output Pressure KW Usage If you select to display one or more of these results, enter the number of digits to display to the right of the decimal.

Optional Specifications: Keyword Input

Click Keyword Input. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the "C-*" before it and enter the units in parenthesis and any values after the =. Any line beginning with "*" will not be written to the DESIGN II input file. However any line with a "C-" (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this dialog. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This dialog provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Pump Examples

There are several sample flowsheets in "Chapter 38: Pump Samples - c:\designii\samples\equipmnt\pump" of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.34: Reactor

General

The Stoichiometric Reactor module calculates component distribution for the conditions existing at equilibrium, the heat released or absorbed during the reaction, and the resultant temperature changes.

Details

The Stoichiometric Reactor module calculates component distribution for the conditions existing at equilibrium, the heat released or absorbed during the reaction, and the resultant temperature changes. The temperature and enthalpy of the product are calculated based on stoichiometry.

Exactly one inlet and one outlet stream must be connected to the reactor. Any snap point on the reactor symbol may be used for the purpose. The inlet stream may be two phases. To separate the phases of the outlet stream you must use the Flash module.

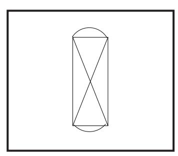
NOTE: If the temperature out of the Reactor will be in excess of 1000 degrees F, you should consider using Vapor Pressure equilibrium K-values and entering ideal vapor heat capacities and latent heat, or entering tabular K-values and enthalpies via a Chemical File (requires the ChemTran program). Equilibrium K-values and Vapor/Liquid Enthalpy choices may be entered on the Basic Thermo dialog. A Chemical File Name may be entered on the Advanced Thermo dialog.

Please see the online **DESIGN II** Help, topic ***Equipment/Reactor*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 32: Reactor*** for more details.

Symbols

The Reactor unit module has one symbol:

You must connect one inlet and one outlet stream to the Reactor module.



Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

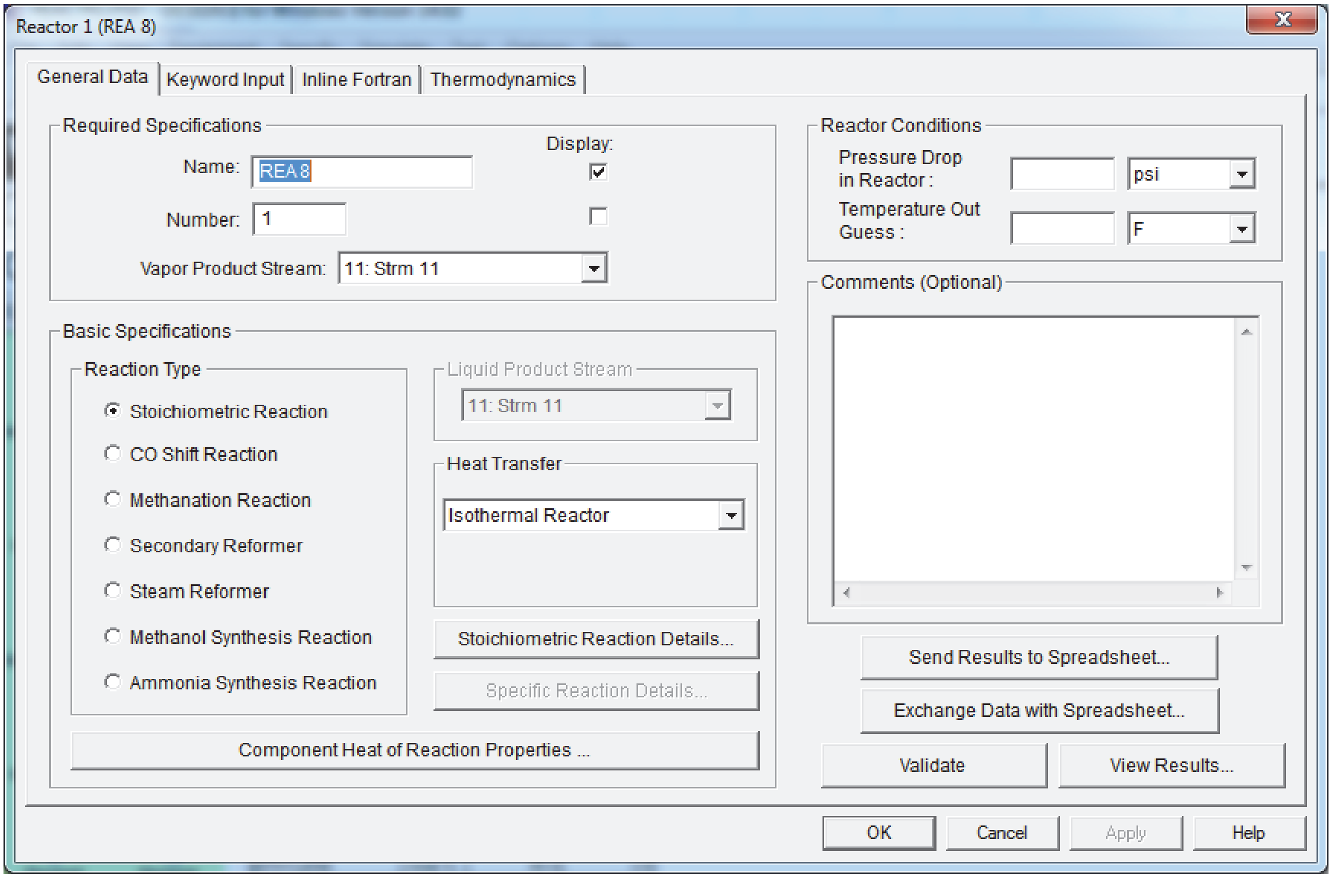


Figure 1: Stoichiometric Reactor Dialog (from reactr1.psd)

General Data

Data Items	Description
Name/Number Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Vapor Product Stream	Open the drop down list and select the desired stream to designate as the Vapor Product Stream.
Liquid Product Stream	Open the drop down list and select the desired stream to designate as the Liquid Product Stream.
Reactor Conditions	
Pressure Drop in Reactor	Enter the change in pressure from the inlet to the outlet and select a unit.
Temperature Guess Out	Enter an estimated temperature for the reactor products and select a unit.
Heat Transfer	
Adiabatic reactor	Product stream enthalpy is equal to the feed stream enthalpy plus the heat of reaction, and the temperature of reactor product is calculated from the product composition and enthalpy. This is the default type for a reactor module and is an option for an equilibrium reactor module.
Isothermal reactor	This is the default type for an Equilibrium Reactor. However, this command is available for a reactor module instead. For the ISOthermal reactor, the temperature of the feed is used as the temperature of the product.
Temperature out	For the TEMperature OUT reactor, the specified product temperature is used with the product composition in calculating the product enthalpy. An enthalpy balance will be performed to calculate the amount of heat added to or removed from the reactor to meet the desired temperature.

Reactor Duty	Enter the total duty for the reactor. The temperature of reactor products is calculated.
Reaction Type	
Stoichiometric Reaction	The stoichiometric reactor type calculates component distribution using key component conversion and stoichiometric numbers for products and reactants.
CO Shift Reaction	The reaction involved is CO(g) + H2O (g) = CO2(g) + H2(g) Methane, as well as other hydrocarbon components, will be treated as inert and will not affect the calculation. Oxygen cannot be present in the feed to this reaction. Calculations will stop if it is present.
Methanation reaction	The reactions are: CO(g) + H2O(g) = CO2(g) + H2(g) CO(g) + 3H2 (g) = CH4(g) + H2O(g) Hydrocarbons other than methane will be considered as chemically inert but their presence will affect the reaction calculation as the second equation is not equimolar. Oxygen cannot be present in the feed or the calculation will stop.
Secondary Reformer	The reactions are the same as the METHANator but oxygen is allowed and is totally consumed producing CO, CO2, H2O, and CH4 coexisting in equilibrium. Hydrocarbons other than methane will be considered as chemically inert. However, their presence will affect the reaction calculation as the second equation is not equimolar.
Steam Reformer	Hydrocarbons and any of the following components are allowed to react: CO, CO2, H2, H2O, and O2. The components will react and achieve thermal and mass equilibrium as in the Methanator. Any oxygen found in the input stream will be totally consumed. NOTE: You are allowed to override the default heating values for petroleum fraction components involved in STEam REFormer calculations.
Methanol Synthesis reaction	The reactions are: CO(g) + H2O(g) = CO2(g) + H2(g) CO(g) + 2H2 (g) = CH3OH(g) Components other than methanol are allowed and will be treated as chemically inert. However, they will affect the reaction calculation as the second equation is not equimolar.
Ammonia Synthesis reaction	The reaction is: N2(g) + 3H2(g) = 2NH3(g) Other components are allowed but they will be treated as chemically inert. However, their presence will affect the reaction calculation as it is not equimolar.
Stoichiometric Reaction Details	If you selected Stoichiometric Reaction, click this button to display the Stoichiometric dialog.
Specific Reaction Details	If you selected any Reaction Type EXCEPT **Stoichiometric Reaction**, click this button to display the Specific Reaction dialog.
Component Heat of Reaction Details	Click this button to display the Component Heat of Reaction Properties tab on the Thermodynamic and Transport Methods dialog.

Specific Reaction Details

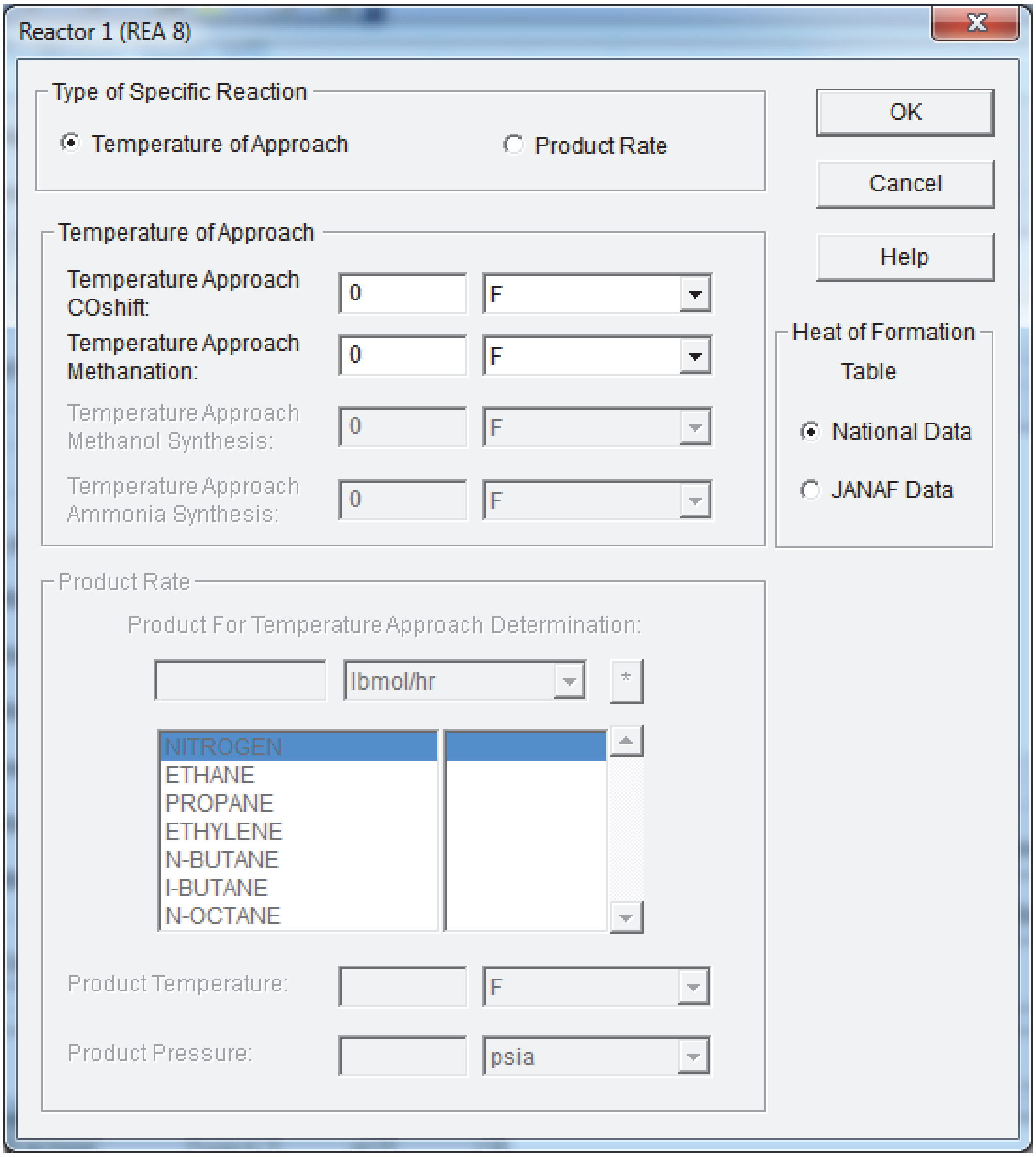


Figure 2: Reaction Dialog (from reactr1.psd)

You should determine if the default temperature of approach value is reasonable for your simulation. If it is not, you can enter appropriate values for the temperature of approach for specific reactions. The other approach is to use the PROducts commands to describe the behavior of the equilibrium reactor.

The outlet temperature, pressure and composition can be obtained from plant data or catalyst vendor information. These conditions are used only to determine a temperature of approach for a particular reactor and catalyst. They do not necessarily have to be identical to the conditions in a DESIGN II simulation of this reactor.

Data Items	Description
Temperature Approach CO shift	Enter the desired temperature of approach for the COSHIFT reaction. Can be used for all reactions except AMMonia SYNthesis. Default is 0 F.
Temperature Approach Methanation	Enter the desired temperature of approach for the Methanation reaction. Can be used with reactions for METHANation, SECondary REFormer, and STEam REFormer. Default is 0 F.
Temperature Approach Methanol Synthesis	Enter the desired temperature of approach for the METhanol SYNthesis reaction. Only used with the METhanol SYNthesis reaction. Default is 0 F.
Temperature Approach Ammonia Synthesis	Enter the desired temperature of approach for the AMMonia SYNthesis reaction. Only used with the AMMonia SYNthesis reaction. Default is 0 F.

NOTE: Temperature of approach (of a certain reaction) is defined as the difference between the actual exit stream temperature of the actual reactor and the temperature at which the reactor product composition would be in equilibrium. The temperature difference can sometimes have a negative value.

Product Rate Specifications

The temperature of approach can also be determined by using the three PROduct commands below.

Data Items	Description
Product for Temperature Approach Determination	Enter the product stream compositions in the same order as the COMponent list in the GENeral section. DESIGN II will determine the temperature deviations from the product temperature.
Product Temperature	Enter the product temperature.
Product Pressure	Enter the product pressure.

Heat of Formation Table Commands

For all the specific reaction types, you should choose either the JANAF Thermochemical Tables or the NATional Bureau of Standards data to describe the chemical equilibria. This command is not valid for the stoichiometric reaction type.
(select only one)

Data Items	Description
National data	National Bureau of Standards Thermochemical Table is used to calculate heats of formation. This is the default.
Janaf data	Joint Army, Navy,and AirForce Thermochemical Table is used to calculate heats of formation.

NOTE: Operating temperature for each reactor should not exceed 3800 F (2093.3 C) if you use JANAF data or 2200 F (1204.4 C) if you use National Bureau of Standards data.

Stoichiometric Reaction Details

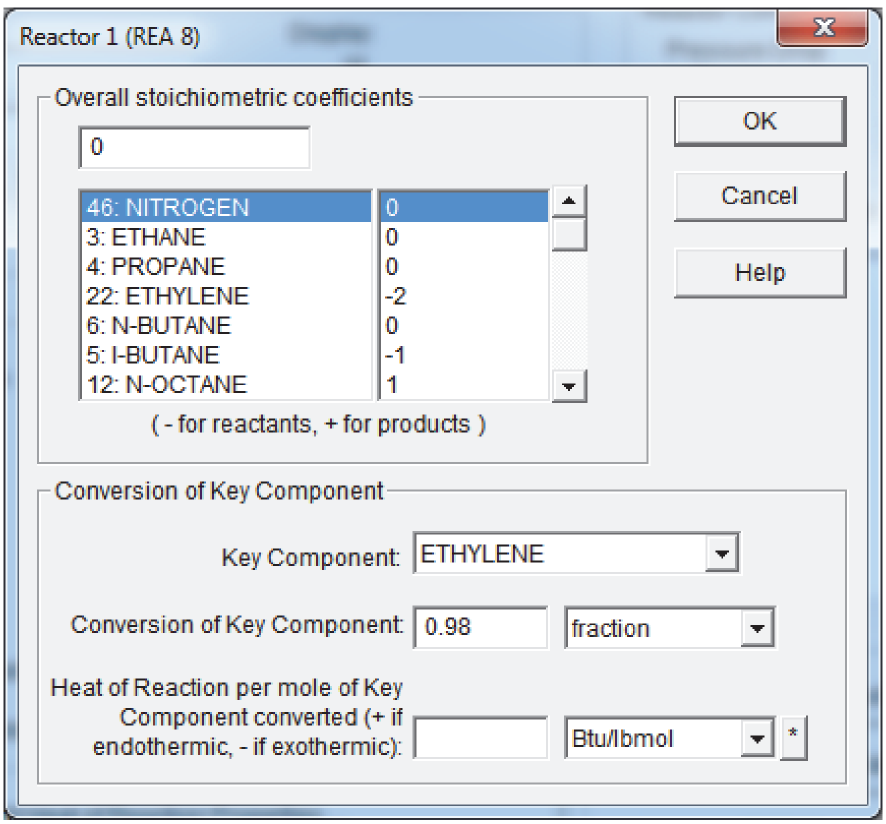


Figure 3: Stoichiometric (from reactr1.psd)

Data Items	Description
Limiting reactant	Enter the component ID number for key component.
Conversion	Enter fractional conversion of key component.
Stoichiometric Coefficients	The Stoichiometric Coefficients for the reactants (enter as negative numbers) and the products (enter as positive numbers) are specified. For components not affected by the

reaction, Stoichiometric Coefficients are entered as zero. All coefficients for the components must be coded in the same order as their component ID numbers on the component list.

Note: Fractional stoichiometric coefficients may be entered.

Heat of reaction — Enter the heat of reaction per mole of key component converted. Enter a negative number if the reaction is exothermic. This heat of reaction is based on ideal gas at 32 F and 1 atmosphere. This command is optional but highly recommended for the simple reactor.

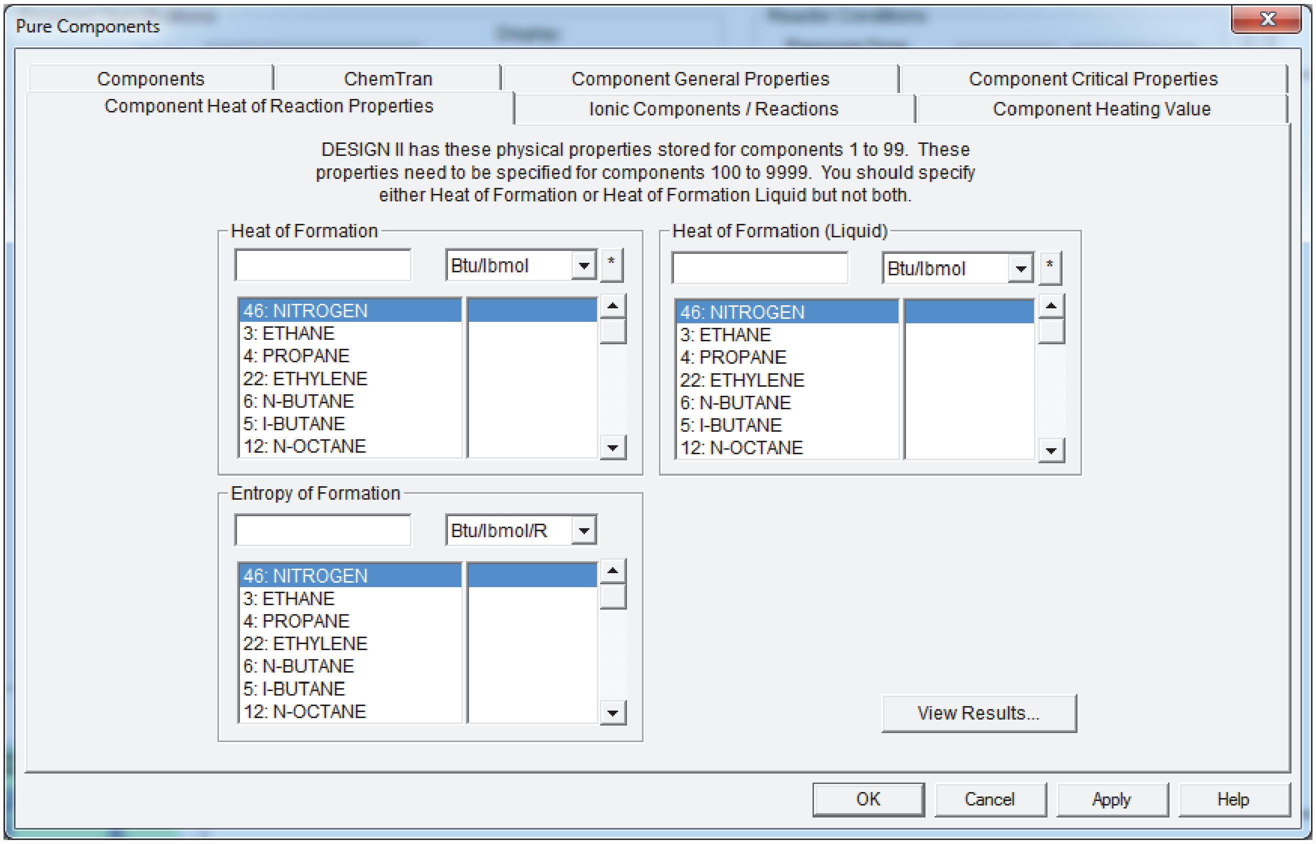


Figure 4: Component Heat of Reaction Properties tab (from reactr1.psd)

Heat of Reaction Properties

For reactors using specific reactions, Heat of Formation at 25C must be supplied for every component with an ID number greater than 99.

Data Items	Description
Heat of Formation i	Enter the heat of formation where i is the component ID number.
OR	
Heat of Formation (liquid)	Enter the heat of formation on a liquid basis for the component.
Entropy of Formation i	Enter the entropy of formation where i is the component ID number.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the "C-*" before it and enter the units in parenthesis and any values after the =. Any line beginning with "*" will not be written to the DESIGN II input file. However any line with a "C-" (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Reactor Examples

There are several sample flowsheets in “Chapter 39: Reactor Samples - c:\designii\samples\equipmnt\reactor” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.35: Refine Column

General

Use the Refine module to rigorously model distillation columns receiving high hydrocarbon-content feeds. The column configurations include condenser (partial or total), pumparound, and pumparound with a condenser (partial or total). You may add sidedraws, side strippers, heaters, coolers and feeds to these configurations.

Details

The Refine module is used for rigorous simulation of distillation columns receiving high hydrocarbon-content feeds. Many common refinery units can be modeled, including preflash towers, atmospheric columns, vacuum columns, FCC main fractionators and quench columns. Side strippers, pumparounds, and feed furnaces can be added to the main column configuration for a rigorous, simultaneous calculation.

This module can be used for designing, modifying refinery fractionators with very good results. It is possible to duplicate the operation of existing units within a few percent of actual plant data.

Many important operational parameters can be matched, such as tower temperatures, reflux, product rates, and property specifications. Significant discrepancies are usually due to inaccuracies in data collection but may be due to operational problems. In these cases, Refine may be used to identify the source of the problem and determine the solution.

Five major column types encompass all common refinery fractionators distinguished by reflux generation and condenser type. All associated operations (reboil, stripping, pumparounds, etc.) are available within each column and multi-column operations are permitted.

The Refine module offers a great deal of flexibility in the definition of column parameters. You are allowed to select essentially an unlimited number of stages, side strippers, pumparounds, heaters/coolers, products, feeds to the main column, steam feeds, feed furnaces, water decants, and components. Refine uses theoretical trays, i.e. equilibrium stages; therefore when simulating existing plants, it is necessary to estimate tray efficiency in each column section to determine the number of equilibrium stages for the model.

The Refine column requires that there be at least one crude stream defined in the flowsheet.

The following topics provide background on the column setup:
- Product Specification And Column Control
- Calculation Techniques
- Water K-values And Decant
- Stream Order Convention
- Tray Numbering Convention
- Step-by-Step Procedure

Product Specification And Column Control
The flexibility of the Refine module allows you to control the fractionation process in several ways; you can:

* Directly specify product rates or qualities such as end points, letting the program adjust product rates to meet your specifications.
* Use FIX commands to set column temperatures, vapor rates, and liquid rates to desired values by varying other parameters.

You can create additional specifications with Inline Fortran For instructions about the use of Inline Fortran; please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Calculation Techniques
The Refine module uses a unique mathematical approach which simultaneously calculates heat/material balance for a main column, side strippers, and pumparounds. By treating these separate units as if they were a single column, the rigorous calculations are completed significantly faster than other available crude unit programs.

In addition, accurate results can be achieved using data that is typically available to you because the Refine module has a great deal of flexibility in specifications and column description.

Water K-values And Decant
The Refine calculation technique handles water in a completely rigorous manner with water automatically decanted at the condenser.

For quench or other columns where several trays may have large amounts of free water present, water decants can be added as needed; water decants are handled as special sidedraws. The pumparounds can be pure water or water-hydrocarbons. Water solubility in hydrocarbons is taken into account for each product.

Stream Order Convention
Certain conventions must be followed when defining feed and product streams in the Refine module. These conventions are:

- Each feed stream to the Refine module is assigned a unique number. Steam feeds are NOT numbered streams. You must connect a stream number on the flowsheet to a feed stream name, using the Feeds dialog. On the DESIGN II keyword command defining feed stream topology, the feed streams will be listed by DESIGN II for Windows in top-to-bottom order.
- Each product stream is assigned a unique number. The product stream numbers are given negative signs and listed in top-to-bottom order on the DESIGN II keyword command generated by DESIGN II for Windows. A decant stream from the condenser is NOT a numbered product stream but any water decant streams from trays are numbered product streams.
- Internal streams, such as pumparound draw and return streams, side stripper draw and return streams, reboiler streams, or reflux streams are not numbered. Their compositions, flow rates, and other properties are automatically estimated and calculated values are reported in the column results.

Tray Numbering Convention

Certain conventions must be followed when numbering trays in the Refine module. In Refine, there is a distinction between "stages" and "trays". The total number of "stages" for the main column are specified, followed by separate values for each side stripper. Tray numbers are used to locate feeds, sidedraws, side heater/coolers, pumparounds, steam feeds, and side strippers.

"Stages" refers to the total number of theoretical trays, including partial condenser and reboilers, if present.

"Trays" are theoretical trays and reboilers. All locations refer to tray locations. Number the trays (including any reboilers) in top-to-bottom order starting with the main column. Tray 1 is the top tray of the main column; tray 0 is the condenser, if present.

Step-by-Step Procedure

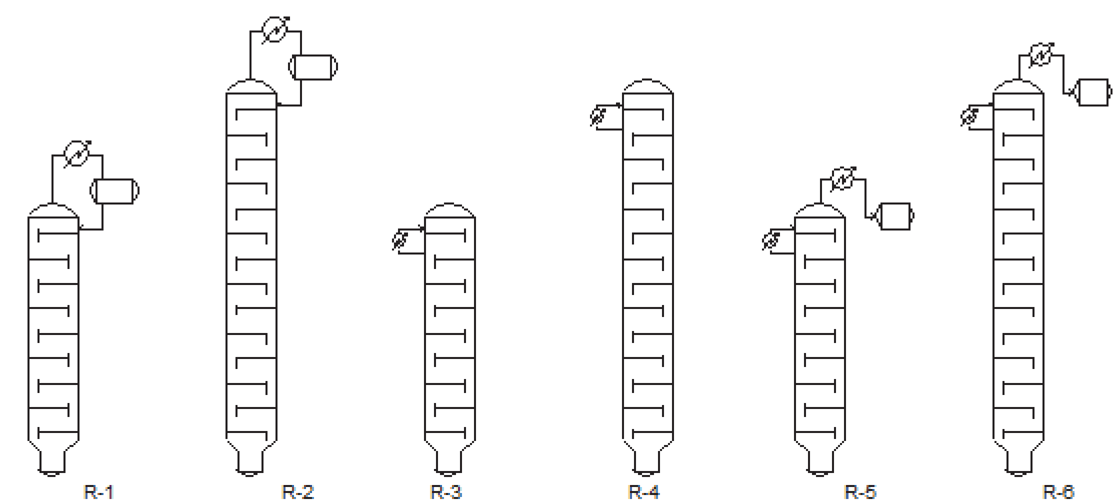
Specifying a Refine column simulation with DESIGN II is simplified by following a step-by-step procedure:

- Select the Refine column type by selecting the combination of condenser and pumparound you want from the Refine 1, Refine 2, or Refine 3 shapes. The shapes are displayed in the Browser.
- By selecting the dialogs under Required Specs, add any required column configuration specifications, including heat and product specifications, and feed description.
- Add any sidedraws, heaters/coolers, pumparounds, and water decants using the Optional Specs dialogs.
- Add heat sources (steam, reboiler, or vapor feed); at least one heat source is required at the bottom of the column.
- Add any pumparounds and associated specifications.
- Add optional tray fixes and product property specifications.
- Add optional commands as needed for cooling curves, print control, column sizing and more.

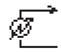
Please see the online **DESIGN II** Help, topic ***Equipment/Refine Column*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 33: Refine Column*** for more details.

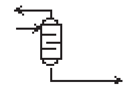
Symbols

The Refine Column unit module has three symbols: Refine 1 (condenser), Refine 2 (pumparound), Refine 3 (condenser and pumparound). There are two sizes for each symbol, regular and tall.



The Refine module requires that you connect at least one inlet stream (feed) and two outlet streams (top and bottom products) to the module. The top and bottom outlet streams must be connected to the top and bottom snap points (at the top and bottom of the column). If there is a condenser present on the column; then there must be an outlet stream connected to the snap point on the bottom of the condenser (top liquid product). If the condenser is a partial condenser; then there must be outlet streams connected to both snap points on the condenser (top vapor product and top liquid product). If the condenser is a total condenser, you must connect the top outlet stream to the snap point on the bottom of the condenser.





There are also 3 add-on symbols to help present the refine columns: a pumparound symbol, a side stripper with reboiler symbol and a side stripper symbol. Each of the side stripper add-on symbols must have one outlet stream attached to them (the outlet stream will be automatically attached to the refine column that the side stripper is connected to).

The number of sidedraws, side strippers, and inlet streams are limited to the total number of available snap points. There must be an outlet stream connected to the snap points on the sides of the Refine symbol for each sidedraw and side stripper. The number of outlet streams (top and bottom products, side strippers and sidedraws) is limited to 12. To determine the number of inlet streams you can connect; subtract the total number of side strippers and sidedraws from 18 for a column without a condenser; and 17 for a column with a condenser.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

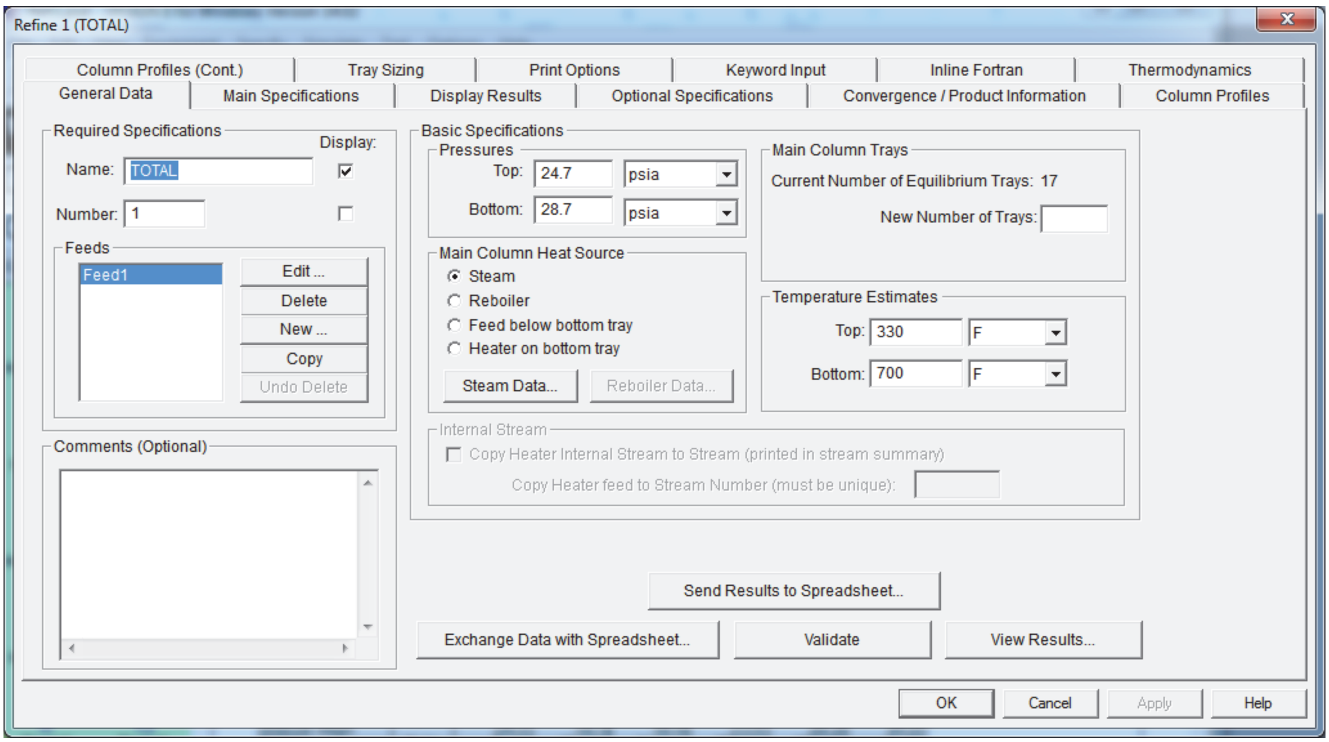


Figure 1: Refine Dialog (from refi1.psd)

General Data

Enter the basic specifications for Refine Columns using this dialog, which is organized into group boxes and cascading dialog buttons. Enter or select information for all fields on this dialog.

Data Item	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment, ; then choose to display it on the flowsheet.
Feeds	Enter feeds to use. Click the New button to display the Feed - Specifications dialog that you can use to create new feed specifications. Select an existing feed and click the Edit button to modify the feed specifications. Select an existing feed and click the Delete button to remove that feed. Select an existing feed and click the Copy button to replicate the feed. You can then use the Edit function to make any minor changes to the copied feed.

Display Results on Flowsheet	You can optionally display the following results on the flowsheet: Trays Reflux Ratio Top Temperature Condenser Duty Bottom Temperature Condenser Duty Reboiler Duty If you select to display one or more of these results, enter the number of digits to display to the right of the decimal.

Pressures

Use this group box to enter pressure specifications for the column; using these specifications, DESIGN II generates a linear pressure profile for the trays. If you have a nonlinear profile, select Profiles from the Optional Specs menu.
You must use the same pressure dimensional units for the main column and the side strippers.

Data Item	Description
Top	Enter the top tray pressure. If you have a nonlinear pressure profile, the complete Pressure Profile (a pressure for every tray) can be entered (see the Profiles/Pressures dialog).
Bottom	Enter the bottom tray pressure for the column.

Main Column Trays

Use this group box to enter the number of trays in the column (not including the condenser and reboiler, if present).

The top tray is always tray 1. The trays are numbered from top-to-bottom. Do not count the condenser or reboiler in the number of trays. DESIGN II numbers the partial condenser as tray zero and the reboiler as one greater than the number of trays in the column. A minimum of two trays is required in the main column.

Data Item	Description
Current Number of Trays	Reports to you the number of trays you have specified for the column; if you have not entered the number of trays, the space will be blank.
New Number of Trays	Enter the number of trays for the column. If you change the number of trays in the column, you may need to make adjustments in other specifications, such as the location of sidedraws or feeds. When you change the number of trays a third edit box appears, which allows you to add or delete trays in a particular section of the column.

Main Column Heat Source

Use this group box to select the heat source for the main column. The heat source is either a steam feed below the bottom tray, a reboiler below the bottom tray, a vapor feed below the bottom tray, or a heater on the bottom tray. The default heat source is a steam feed below the bottom tray.

If the heat source is steam, you must click the steam button to bring up the steam data dialog; you can then specify steam data.

If the heat source is reboiler, you must click the reboiler button to bring up the reboiler data dialog; you can then specify reboiler data.

If the heat source is either vapor feed or a heater, specify these using the Specs menu on the Refine data window. The menu paths are Specs/Feeds... or Specs/Heaters/Coolers.

If the heat source is a heater then there must be a heater on the bottom tray. If the heat source is a vapor feed then there must be a feed below the bottom tray.

The DESIGN II commands generated for the main column heat source are specified on the following dialogs:

- Steam
- Reboiler
- Vapor feed stream below the bottom tray
- Tray heater on the bottom tray

Data Item	Description
Steam	Select this radio button if you want a steam feed below the bottom tray for the main column heat source.
Reboiler	Select this radio button if you want a reboiler below the bottom tray for the main column heat source.
Vapor feed below bottom tray	Select this radio button if you want a vapor feed stream below the bottom tray for the main column heat source.
Heater on bottom tray	Select this radio button if you want a heater on the bottom tray for the main column heat source.

Temperature Estimates

Use this group box to enter estimates for the temperatures at the top and bottom of the column. DESIGN II estimates a linear initial column profile from these temperature guesses for most columns.

In some cases, where the temperature profile is far from linear, it may be better to estimate the top or bottom temperature different from the actual temperature. Specifically, addition (or removal) of heat to tray(s) with side heaters/coolers can cause a nonlinear temperature profile.

In cases where nonlinear column temperature profiles are expected, a temperature profile for every tray in the column can be entered and may be required for convergence (see the optional Temperature dialog under Specs/Optional/Profiles).

You must use the same temperature dimensional units for the main column and the side strippers.

NOTE: For PUMREF with FIX TEM on top tray specifications, enter an estimate for TEMTOP on Tray 2.

Data Item	Description
Top	Enter the top product temperature as an initial guess. If the Temperature of Condenser specification is also entered; then this entry should be the top tray temperature.
Bottom	Enter the bottom product temperature as an initial guess.
Internal Streams	
Copy Heater Internal Streams to Streams	Select this checkbox to add internal stream data for the heater to a stream, so that the data will appear in the stream summary output. Copy Heater Internal feed to Stream Number (must be unique): Enter a stream number. Copy Heater Internal return to Stream Number (must be unique): Enter a stream number.

Required Specifications: Feeds

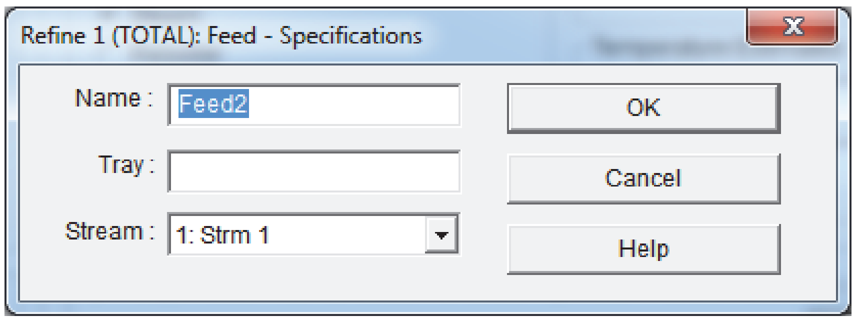


Figure 2: Feed – Main (from refi4.psd)

You can assign flowsheet streams as feeds to the distillation/Refine column and assign the feed to a column tray.

This dialog is used to enter the name, tray location, and stream identity for column feeds. The data entries are described in the table below.

Feeds can enter on any theoretical tray in the column. The top tray is always tray one and the condenser, if present, is tray zero. Any liquid in the feed will go on the feed tray. This means if the feed is two phase at the tray pressure; then the liquid portion of the feed enters the feed tray and the vapor portion automatically mixes with the liquid in the tray above. If the feed is all vapor; then all of the feed enters and mixes with the liquid in the tray above the feed tray.

Feeds are always adiabatically flashed at the tray pressure. If the feed pressure is less than the tray pressure, calculations will continue but a warning message will state that a feed pump is required.

Data Item	Description
Name	Enter the name of the feed. If no entry is made, the default name that appears here will be used for the feed name.
Tray	Enter the tray location for the feed. The top tray in the column is always tray number one.
Stream	This is a combo box that allows you to choose which of the flowsheet streams represents the feed. When you select the Stream combo box, a list of flowsheet streams will appear that you have created for the flowsheet. If you select one of these streams, its name will appear at the top of the box. Alternatively, you can enter the stream name directly in the edit portion of the combo box.

Steam Data

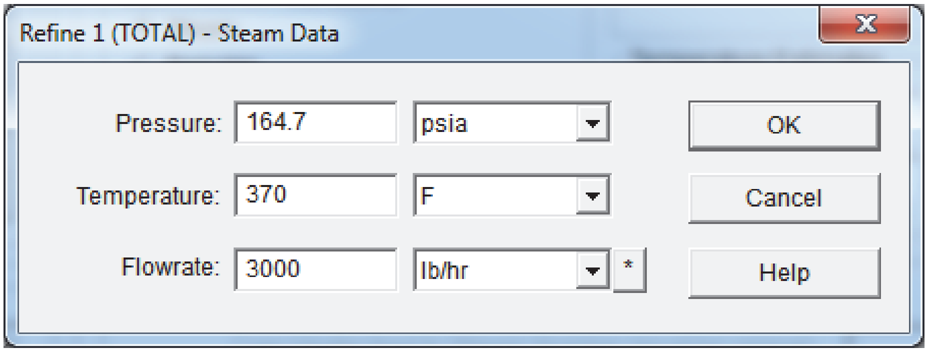


Figure 3: Steam Dialog (from refi1.psd)

Use this dialog to enter the steam feed specifications for the main column or side strippers. The steam feed is always located below the bottom tray in the column or the side stripper. Any additional steam feeds must be handled as separate feeds to the column using Specs/Feeds.

DESIGN II will always calculate the steam feed location for you. Enter the steam feed pressure, temperature, and flowrate in the appropriate boxes.

You must make all dimensional unit choices for pressure, temperature, and flowrates consistent for the main column and each side stripper (e.g. PRE STE (ATM)= 10,10,10)

The steam feed to column or side stripper is always below the bottom tray; this is automatically calculated for you.

Data Item	Description
Pressure	Enter the steam feed pressure.
Temperature	Enter the steam feed temperature.
Flowrate	Enter the steam feed flowrate.

Reboiler Data

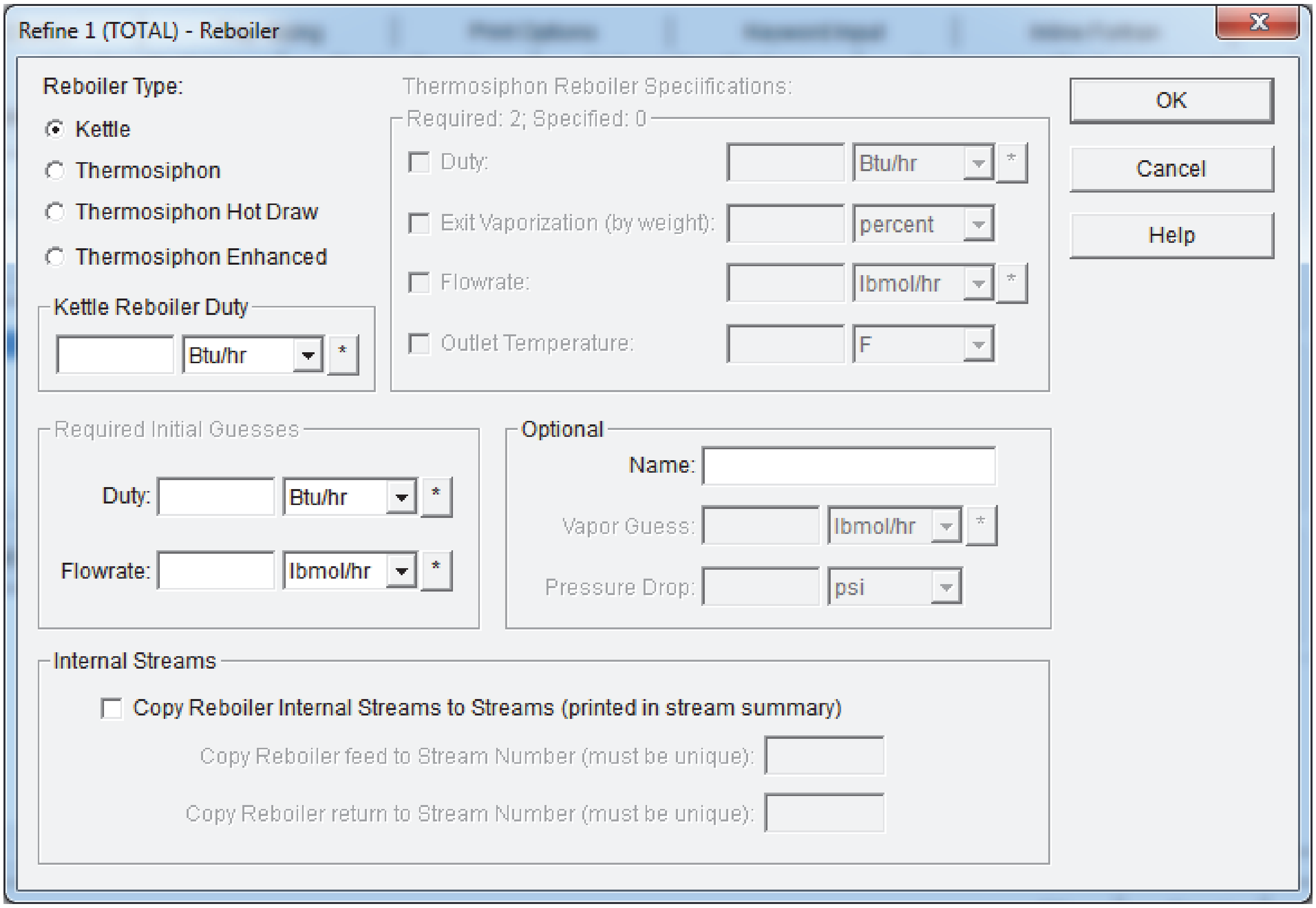


Figure 4: Reboiler Dialog (from refi1.psd)

Use this dialog to enter data describing a reboiler for the main column or a side stripper. You may select one of four types of reboilers: kettle, thermosiphon, thermosiphon hot draw or thermosiphon enhanced.

The default reboiler type is kettle; the enhanced thermosiphon reboiler includes the vaporization effects of mixing the sump liquid with the reboiler return liquid.

If the reboiler type is a kettle, you must enter a duty.

Thermosiphon reboilers require two specifications from the list below:

- Duty
- Exit vaporization percent
- Flowrate
- Outlet Temperature

You must enter a guess for reboiler duty and/or flowrate if you did not choose them as specifications. You may enter guesses for the vapor flow from the reboiler (on a molar basis) and for the pressure drop (rise) between the bottom tray and the reboiler outlet. If you do not enter a guess for the pressure drop, the value you entered for Bottom Tray Pressure will be used.

Data Item	Description
Required Initial Guess	
Duty	Enter the reboiler duty and select a unit.
Flowrate	Enter the reboiler flowrate and select a unit.
Optional	
Name	View the name or change it. A name can contain up to 16 characters; however, all the product names combined should not exceed 65 characters.
Vapor Guess	Enter a guess for the vapor flow from the reboiler (on a molar basis).
Pressure Drop	Enter a guess for the pressure drop (rise) between the bottom tray and the reboiler outlet.
Internal Streams	
Copy Reboiler Internal	Select this checkbox to add internal stream data for the reboiler to a stream, so that

Streams to Streams — the data will appear in the stream summary output.
Copy Reboiler feed to Stream Number (must be unique): Enter a stream number.
Copy Reboiler return to Stream Number (must be unique): Enter a stream number.

Required Specifications: Main (Condenser without Pumparound – Refine 1 / 1A symbol)

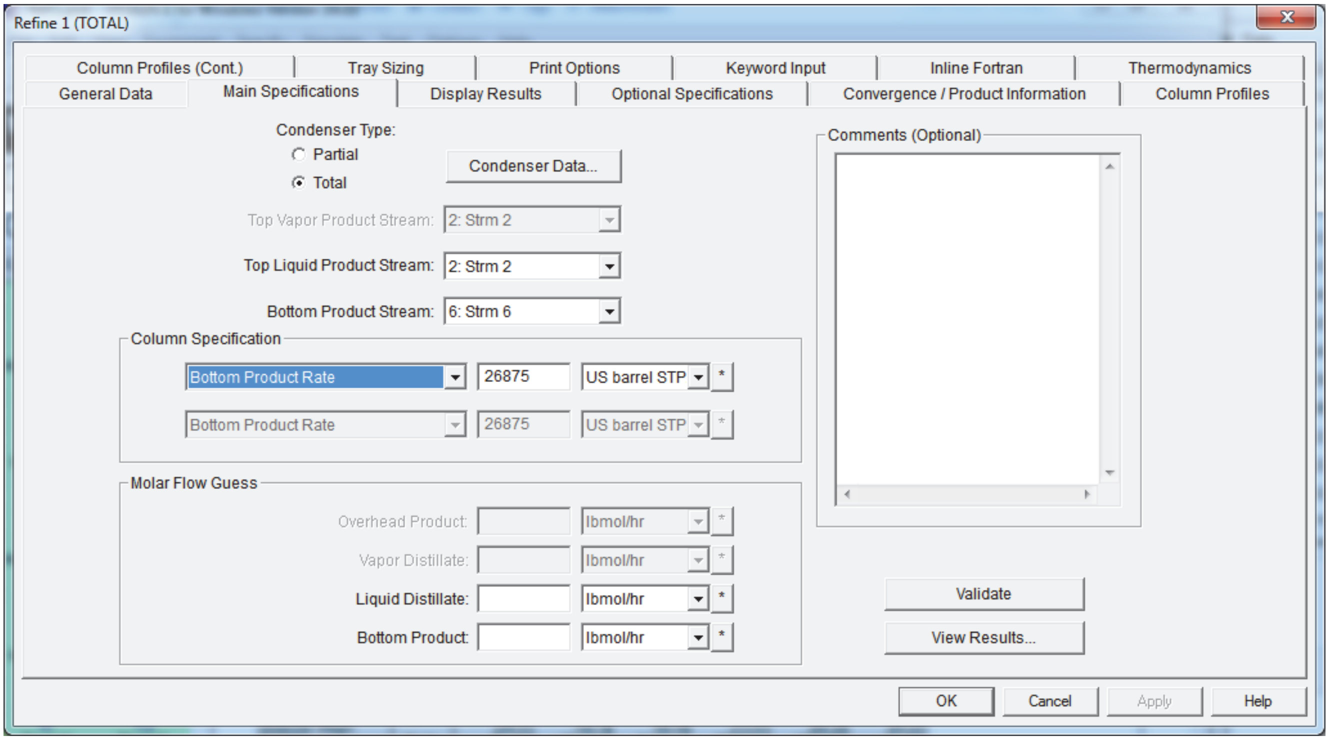


Figure 5: Main Specifications Tab (from refi1.psd)

Enter the main specifications for Refine columns using this tab, organized into group boxes, radio buttons and cascading dialog buttons.

The radio buttons control the condenser type specification; the condenser type can be either partial (default) or total. The condenser type controls the type and number of column specifications. If you select partial; then two column specifications are required; if you select total; then only one column specification is required.

The condenser automatically decants water (if it is present in sufficient quantity) as a separate liquid phase. Solubility of water in liquid hydrocarbons is taken into account for both the liquid distillate and the reflux returning to the column. The water decant is not represented by a flowsheet stream but is included the simulation report.

The partial condenser has a top liquid product and a top vapor product. The total condenser only has a top liquid product. The total condenser does not have a top vapor product.

Top Liquid Product Stream — Open the drop down list and select a stream to designate as the Top Liquid Product Stream.

Bottom Product Stream — Open the drop down list and select a stream to designate as the Bottom Product Stream.

Column Specifications
Use this group box to enter specifications for the column. If the column has a partial condenser; then two specifications are required. Otherwise, the column has a total condenser and only one specification is required.

The specification choices for the total condenser are:
- Condenser Duty
- Top Liquid Product Flowrate
- Bottom Product Flowrate
- Reflux Ratio

NOTE: If you select Condenser Duty or Reflux Ratio; then you must also enter molar guesses for the top liquid product and bottom product.

The partial condenser specification choices are presented in two cascade box lists as shown below. You must select a different choice from each box.

Left Cascade Box List — Right Cascade Box List

Condenser Duty	Top Liquid Product Flowrate
Condenser Temperature	Reflux Ratio
Top Vapor Product Flowrate	Total Distillate Overhead Product Flowrate
Bottom Product Flowrate	Bottom Product Flowrate
	Top Vapor Product Rate

NOTE: If you select Condenser Duty and Reflux Ratio, you must also enter molar guesses for top vapor product, top liquid product, and bottom product. If you select Overhead Product Flowrate; then you must select Condenser Temperature as the other choice. If the dimensional units that you selected for the Overhead Product Flowrate are non-molar, you must enter an Overhead Product Flowrate Guess.

Product flowrate specifications using molar dimensional units are entered on a wet (with water of saturation) basis. The water saturation value will be low typically, except for vapor products. Product flowrate specifications using volumetric or mass units are entered on a dry (water free) basis. All product flowrates must be specified in the same dimensional units.

The reflux ratio is defined as the molar flow of liquid returned to the column divided by the total overhead product (liquid plus vapor), excluding any decanted water.

Data Item	Description
Condenser Duty	Enter the duty for the condenser.
Condenser Temperature	Enter the temperature of the condenser.
Top Vapor Product Flowrate	Enter the flowrate for the top vapor product.
Top Liquid Product Rate	Enter the flowrate for the top liquid product.
Bottom Product Rate	Enter the flowrate for the bottom product.
Total Distillate Overhead Product Flowrate	Enter the flowrate for the combined overhead products.
Reflux Ratio	Enter the reflux ratio for the condenser.

Molar Flow Guess

Use this group box to enter molar product flowrate guesses for the column. Enter molar guesses if you are using product flowrates in mass or volume dimensional units; DESIGN II uses the molar guesses to calculate initial vapor and liquid rates on the trays. Molar guesses are always entered on a dry basis. Use the same dimensional units for all molar product flowrate guesses.

You can only enter the Overhead Product Flowrate Guess if you specified the Overhead Product Flowrate. If the dimensional units you are using for the Overhead Product Flowrate are non-molar; then you must enter the Overhead Product Flowrate.

The Vapor Distillate Flowrate Guess can only be entered for a column with a partial condenser.

Data Item	Description
Overhead Product	Enter the molar flowrate guess for the combined overhead products.
Vapor Distillate	Enter the molar flowrate guess for the top vapor product.
Liquid Distillate	Enter the molar flowrate guess for the top liquid product.
Bottom Product	Enter the molar flowrate guess for the bottom product.

Required Specifications: Main (Condenser without Pumparound – Refine 1 / 1A symbol) Condenser Data

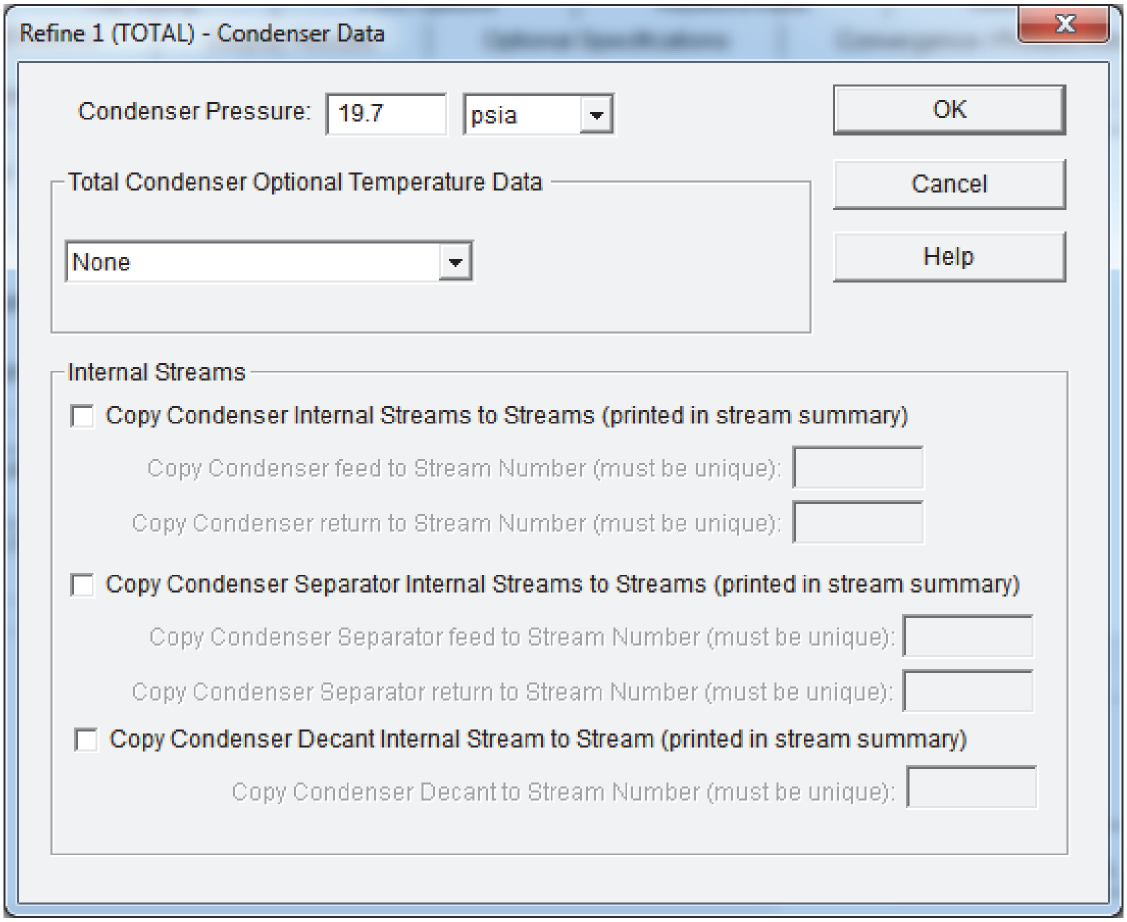


Figure 6: Condenser Data (from refi1.psd)

Use this dialog to enter the specifications for the main column condenser. If you do not specify the condenser pressure; then the default used is the pressure of the top tray in the column.

The default temperature for the total condenser option is the product bubble point. However, you may select one of the following specifications:

- The condenser temperature,
- The degrees of subcooling in the condenser,
- A guess for the condenser temperature.

If the calculated top product bubble point temperature is colder than you guessed or specified, the condenser temperature will be set to the top product bubble point temperature. If you choose the subcooled condenser option, both the top product and the reflux will be subcooled.

For a partial condenser, the condenser temperature specification list will be deactivated.

Data Item	Description
Condenser Pressure	Enter the condenser pressure.
Total Condenser Optional Temperature Data	Enter the condenser temperature.
Internal Streams	
Copy Condenser Internal Streams to Streams:	Select this checkbox to add internal stream data for the pumparound to a stream, so that the data will appear in the stream summary output.
Copy Condenser Separator Internal Streams to Stream Number (must be unique)	Enter a stream number.
Copy Condenser Decant Internal Stream to Stream	Enter a stream number.

Number (must be unique)

Required Specifications: Main (without condenser – Refine 2 / 2A symbol)

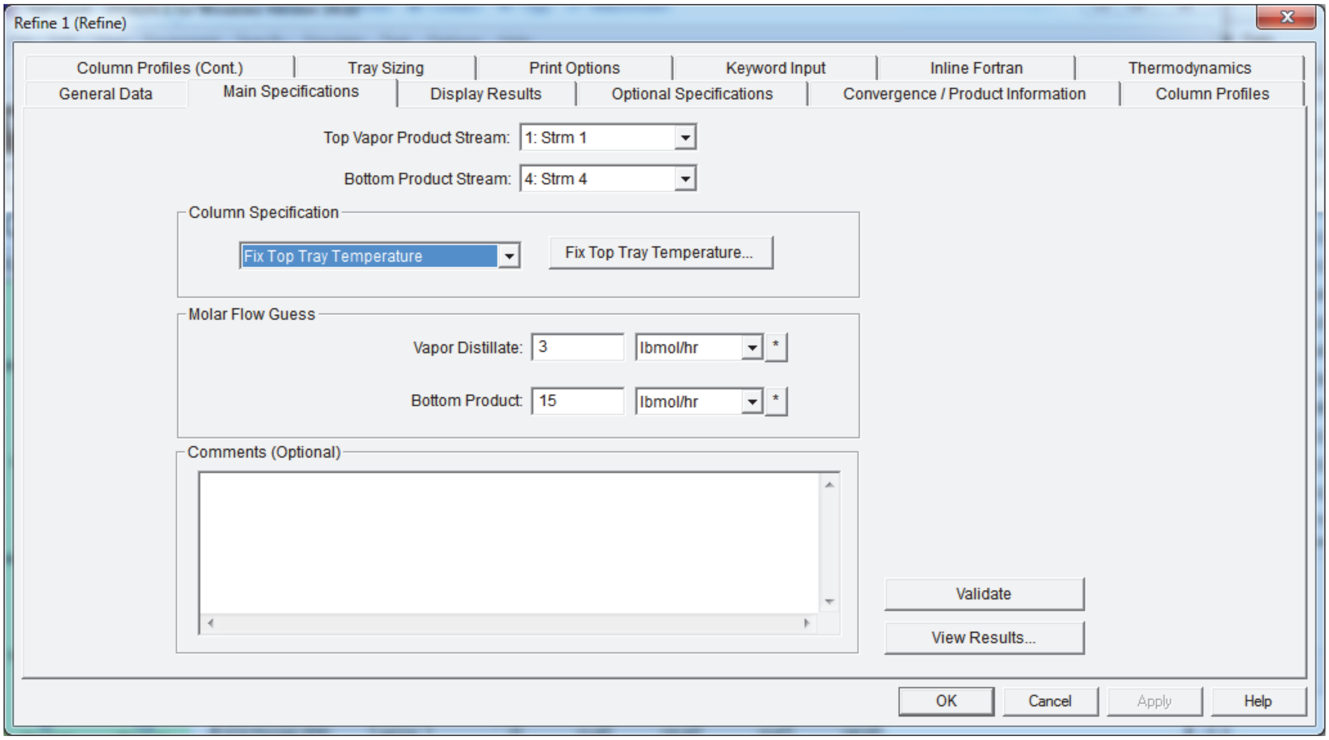


Figure 7: Main Specifications Tab (from refi3.psd)

Enter the main specifications for Refine columns using this dialog, organized into group boxes, radio buttons and cascading dialog buttons.

Top Vapor Product Stream	Open the drop down list and select a stream to designate as the Top Vapor Product Stream.
Bottom Product Stream	Open the drop down list and select a stream to designate as the Bottom Product Stream.

Column Specifications

Use this group box to enter the specification for the main column. The specification choices are:

- Top Vapor Product Flowrate
- Bottom Product Flowrate
- Fix Top Tray Temperature
- Specify Pumparound
- Liquid Feed to Top Tray

Enter product flowrate specifications using molar dimensional units on a wet (with water of saturation) basis. Enter product flowrate specifications using volumetric or mass units on a dry (water free) basis. Specify all product flowrates using the same dimensional units.

The Fix Top Tray Temperature button, Pumparound button, or Specify Feed button is only visible if the specification has been chosen. These button will bring up a dialog for that specification where you can edit a new or existing specification.

For the Fix Top Tray Temperature specification, you must create a Fix Temperature command that sets the temperature value for tray 1. The duty to be varied to achieve this temperature can be numbered tray 1 or higher. For the Pumparound specification, you must define at least one pumparound whose return location is tray 1. The amount of heat removed from this pumparound will be the specification for this Refine column. Both of these specification options require molar guesses for Top Vapor Product and Bottom Product.

NOTE: This Refine column type can be used to model an absorber stripper column. It requires a vapor feed below the bottom tray and a liquid feed to tray 1. Pumparound should be the specification option with HEAt from Pumparound set to 0.0; it requires molar product guesses.

Data Item	Description
Top Vapor Product Flowrate	Enter the flowrate for the top vapor product.

Bottom Product Flowrate	Enter the flowrate for the bottom product.
Fix Top Tray Temperature	Top tray temperature is set by varying a tray duty or product flowrate.
Specify Pumparound	Heat removal from the top tray pumparound.
Liquid Feed to Top Tray	Specify a feed for the top tray.

Molar Flow Guess
Use this group box is used to enter molar flowrate product guesses for the column. Enter molar guesses if you specified the product flowrates in mass or volume dimensional units. The molar guesses will aid the calculation of initial vapor and liquid tray profiles. Always enter molar guesses on a dry (no water of saturation) basis. You must enter all molar product flowrate guesses using the same dimensional units.

Data Item	Description
Bottom Product Flowrate Guess	Enter the molar flowrate guess for the bottom product.
Top Vapor Product Flowrate Guess	Enter the molar flowrate guess for the top vapor product.

Fix Top Tray Temperature

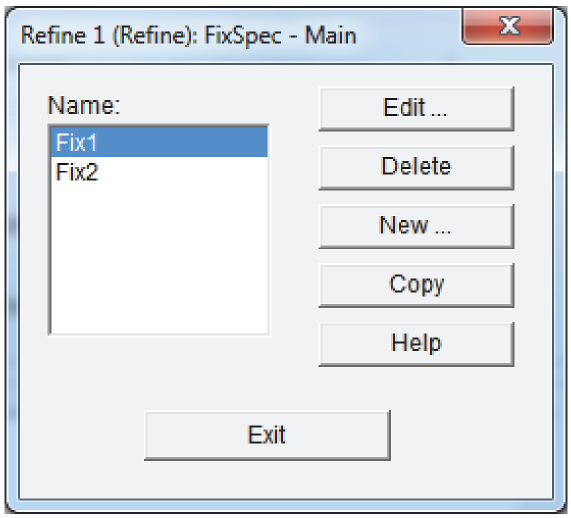


Figure 8: FixSpec - Main Dialog (from refi3.psd)

Use this dialog to assign specifications for the top tray temperature fixes of the Refine column. The dialog consists of a list box called Name and a set of buttons. Use the buttons to perform various operations on the fixes ; two of the buttons display additional dialogs.

The Name list box displays side strippers for the column. DESIGN II for Windows assigns default names. You create a top tray temperature fix and an entry in the list box by selecting the New dialog. If you decide not to use this fix, delete it before leaving this dialog. To delete a fix and its values from the list in Name, select the fix ; then click on the Delete button.

To copy a fix and its values, select the fix ; then click on the Copy button. To change the specifications of an existing fix, click on the Edit dialog button.

When you are finished creating fixes for the column, or editing existing fixes, click on the Exit button.

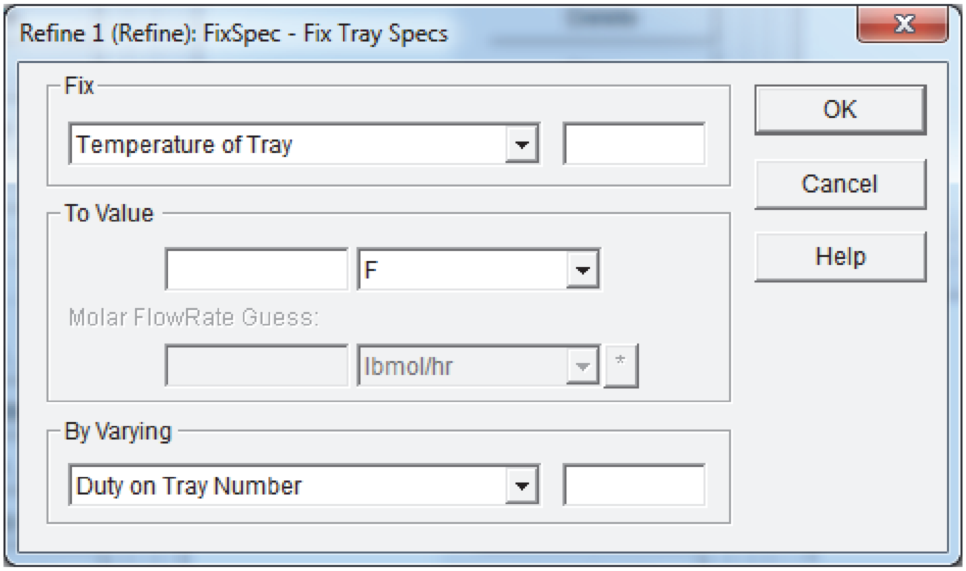


Figure 9: FixSpec – Fix Tray Specs Dialog (from refi3.psd)

Fix Top Tray Temperature

Enter the specifications for fix specifications for trays in the Refine columns using this dialog, organized into group boxes and cascading dialog buttons. You should have an entry or a selection in each of the fields on this dialog.

Fix Tray Specs

This set of commands allows you to fix (or set) an internal tray value which would normally be determined by column calculations by varying a condition which would normally be set (pumparound duty, feed furnace, product rate, reboiler duty, etc.).

Use this group box to enter data for the tray being fixed in the Refine column. You must enter the number of the tray being fixed and select a fix specification. The tray that you fix can be on the main column or on a side stripper. If the tray is on a side stripper; then the tray location is calculated by adding the number of trays in the main column with the number of trays above the tray in the side stripper. The side strippers are ordered from the top to the bottom using the side stripper draw tray as the reference. Add one (1) tray for each reboiler in the main column and reboiler in the side strippers between the main column and the fixed tray.

Use this group box to enter data for the tray being fixed in the Refine column. You must enter the number of the tray being fixed and select a fix specification. The tray that you fix can be on the main column or on a side stripper. If the tray is on a side stripper; then the tray location is calculated by adding the number of trays in the main column with the number of trays above the tray in the side stripper. The side strippers are ordered from the top to the bottom using the side stripper draw tray as the reference. Add one (1) tray for each reboiler in the main column and reboiler in the side strippers between the main column and the fixed tray.

The four fix specifications are: Temperature of Tray, Liquid Flowrate from Tray, Net Liquid Flowrate from Tray, and Vapor Flowrate from Tray. The default fix specification is temperature of the fixed tray.

The most common uses of the fix commands are:

- set top tray temperature by varying the top pumparound duty,
- set flash zone temperature by varying a product draw,
- set overflash by varying a feed furnace,
- set net liquid traffic leaving a tray with both pumparound and side stripper draws by varying a pumparound duty, and
- set reflux flow by varying the main column reboiler.

For example, if you want to fix the reflux from the condenser on the main column and there is a condenser on the main column; then specify the fix tray as 0 (zero). Caution: If you want to fix the temperature of the condenser, use the temperature condenser specification on the Main Specifications dialog; do not fix the temperature of the condenser using the Fix Tray Specs.

At least one tray on the column must remain free of fixes (the temperature of condenser specification counts as a tray fix). You can only select one fix specification per tray. Product flowrates cannot be set using Fix commands; use the product flowrate specification on the side draw dialog, side stripper specification dialog, or main specification dialog.

If the main column reboiler is a thermosiphon reboiler, do not attempt to fix the bottom tray liquid and the reboiler flowrate. If you are varying the duty on the bottom stage of the column; then the reboiler duty has to be a column specification.

Data Item	Description
Fix Tray Specification	Select a tray specification to fix.
Tray to Fix	Enter a tray to be fixed.

Fix Tray Specs To Value
Use this group box to enter the value data for the fixed tray specification in the Refine column. Depending on the specification selected in the fix group box (temperature, liquid flowrate, net liquid flowrate or vapor flowrate), the target value is entered accordingly.

For fix specifications for liquid, net liquid, and vapor which are entered on a non-molar basis, you must enter a molar guess.

Data Item	Description
To Value	Enter the fixed tray temperature, liquid flowrate, net liquid flowrate or vapor flowrate
Molar FlowRate Guess	Enter the flowrate guess if the fix tray value is a flowrate and has non-molar dimensional units

Refine- Fix Tray Specs By Varying
Use this group box to indicate which item is being varied to meet the fix specification in the Refine column: tray duty, feed duty, or product rate. The number of the tray, feed or product being varied must be entered and a vary specification selected. The tray being varied can be on the main column or on a side stripper. If the tray is on a side stripper; then the tray location is calculated by adding the number of trays in the main column with the number of trays above the tray in the side stripper. The side strippers are ordered from the top to the bottom, using the side stripper draw tray as the reference. Add one (1) tray for each reboiler in the main column and reboiler in the side strippers between the main column and the fixed tray.

For example, let's say that you want to vary the third side stripper reboiler duty to fix the vapor rate returning from the third side stripper. The main column has 30 trays and a reboiler. The first side stripper from the top has 4 trays and a reboiler. The second side stripper from the top has 5 trays and no reboiler. The third side stripper has 8 trays and a reboiler. The tray to be fixed is 30 (main column trays) + 1 (main column reboiler) + 4 (first side stripper trays) + 1 (first side stripper reboiler) + 5 (second side stripper trays) + 8 (third side stripper trays) + 1 (third side stripper reboiler = 50.
The three vary specifications are: Duty on a Tray (default), Heater on a Feed, and Flowrate of a Product. The feeds are numbered from the top of the column to the bottom of the column depending on the inlet tray; the products are numbered from the top of the column to the bottom of the column depending on the draw tray from the main column.

If you are varying duty on a tray; then the varied tray must be at or below the location of the fixed tray. Only one variable is allowed per tray. For example, you cannot vary both the feed heater for a feed into tray 19 and the tray duty for tray 19 in order to satisfy two separate fixes.

Data Item	Description
By Varying	Select a specification to vary.
Tray, Feed or Product to Vary	Enter the positional number of the tray, feed or product to be varied.

Specify PumpAround

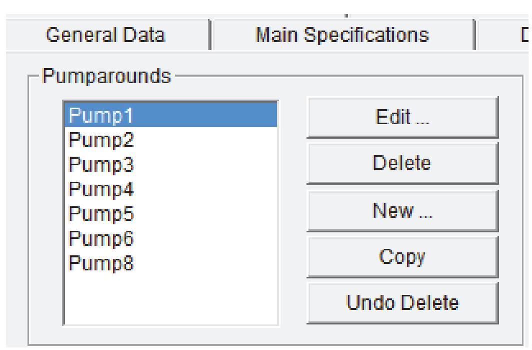


Figure 10: PumpAround - Main Dialog (from refi3.psd)

Use this dialog to assign specifications for the PumpAround of the Refine column. The dialog consists of a list box called Name and a set of buttons. Use the buttons to perform various operations on the PumpAround; two of the buttons display additional dialogs.

The Name list box displays PumpArounds for the column. DESIGN II for Windows assigns default names. You create a PumpAround and an entry in the list box by selecting the New dialog. If you decide not to use this PumpAround, delete it before leaving this dialog. To delete a PumpAround and its values from the list in Name, select the PumpAround ; then click on the Delete button.

To copy a PumpAround and its values, select the PumpAround ; then click on the Copy button. To change the specifications of an existing PumpAround, click on the Edit dialog button.

When you are finished creating PumpArounds for the column, or editing existing PumpArounds, click on the Exit button.

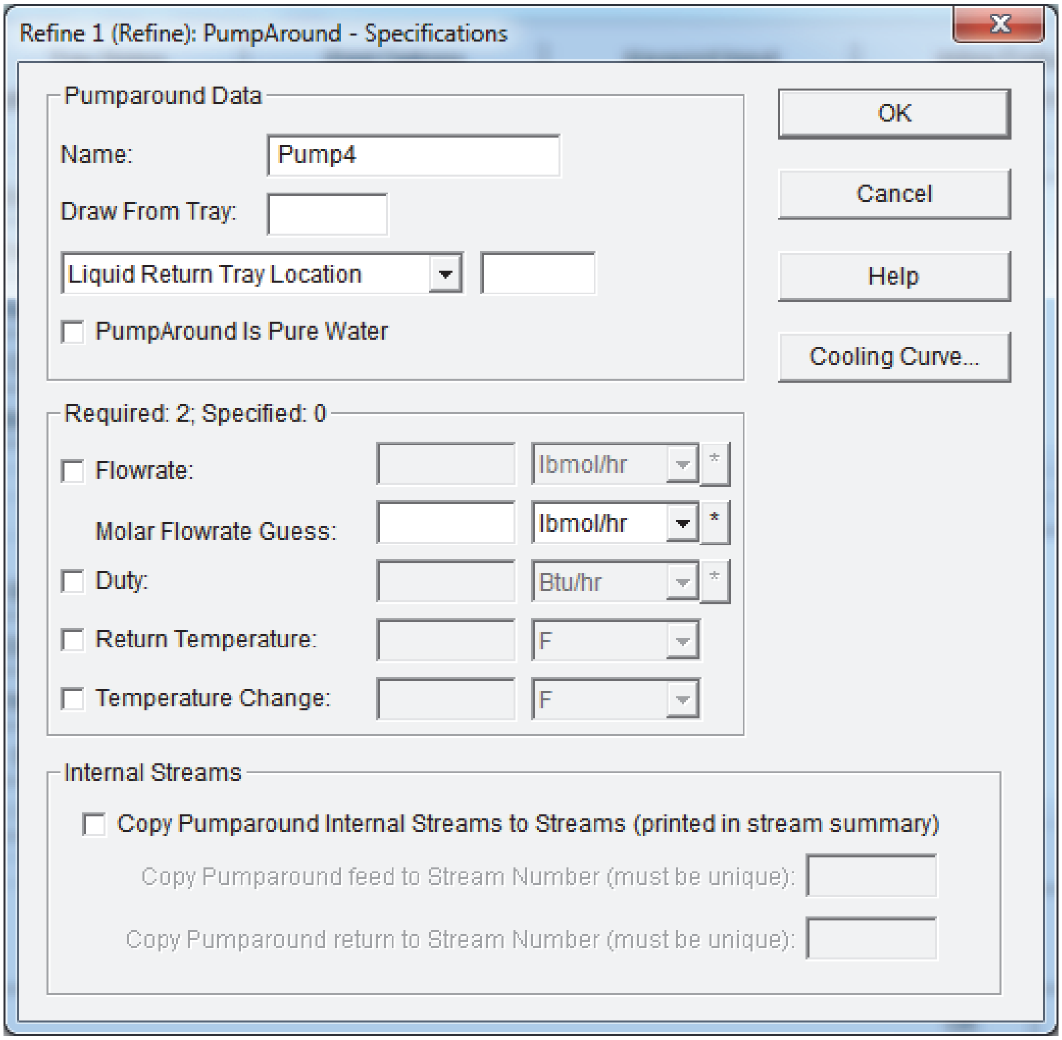


Figure 11: PumpAround - Specifications Dialog (from refi3.psd)

Use this group box to enter specifications for pumparounds attached to the Refine column. The pumparound name is automatically generated but you can change it. You must specify the pumparound draw tray location in the main column.

The Refine column allows for liquids to be pumped up (the return tray is higher than the draw tray), pumped around (the return and draw are on the same tray), and pumped down (the return tray is below the draw tray). Additionally, you are allowed to mix a pumparound with a feed to the main column.

The pumparound can return to a tray in the main column or it can be mixed with a feed to the main column. The default pumparound return is to a tray in the main column. If the pumparound return is to a feed; then the value entered for the return location is the position of the feed in top to bottom order. For example, the top feed is 1, the second feed from the top is 2, and the fourth feed from the top is 4. The return tray number or return feed number must be entered.

If the pumparound contains pure water; then check the Pumparound Is Pure Water box to indicate so. The default composition for pumparounds is a hydrocarbon stream.

NOTE: Water pumparounds require water Partial Solubility profile.

Data Item	Description
Name	View the name or change it. A name can contain up to 16 characters; however, all the product names combined should not exceed 65 characters.
Draw From Tray	Enter the draw tray for the pumparound.
Liquid Return	Select either: Liquid Return Tray Location: Enter the return tray for the pumparound. Liquid Return Feed Number: Enter the feed number with which the pumparound return mixes.
Pumparound Is Pure Water	Check the box if the pumparound is pure water. The default composition for pumparounds is a hydrocarbon stream. **Note**: Water pumparounds require a Water Partial Solubility profile.

Required/Specified — View the number of required specifications and the number already specified. You must select two of the following four specifications for the pumparound: flowrate, duty, return temperature and temperature change. We strongly urge you to use the flowrate of the pumparound liquid as one of the specifications. No heating is allowed for pumparounds.

Flowrate: Click the checkbox; then enter the flowrate of the pumparound liquid and select a unit. To enter base quantity and time units, click the *.

Molar Flowrate Guess: Enter the estimated molar flowrate and select a unit. To enter base quantity and time units, click the *. If the pumparound flowrate has been specified in non-molar dimensional units then the molar flowrate guess must be entered. If the pumparound flowrate was not specified then we recommend that the molar flowrate guess be entered.

Duty: Click the checkbox; then enter the amount of heat removed from the pumparound and select a unit; enter this value as a positive number. To enter base quantity and time units, click the *.

Return Temperature: Click the checkbox; then enter the return temperature of the pumparound and select a unit.

Temperature Change: Click the checkbox; then enter the temperature drop through the pumparound and select a unit.

Internal Streams

Copy Pumparound Internal Streams to Streams: — Select this checkbox to add internal stream data for the pumparound to a stream, so that the data will appear in the stream summary output.

Copy Pumparound feed to Stream Number (must be unique): — Enter a stream number.

Copy Pumparound return to Stream Number (must be unique): — Enter a stream number.

Cooling Curve — Click this button to enter pumparound cooling curve data.

PumpAround Cooling Curve

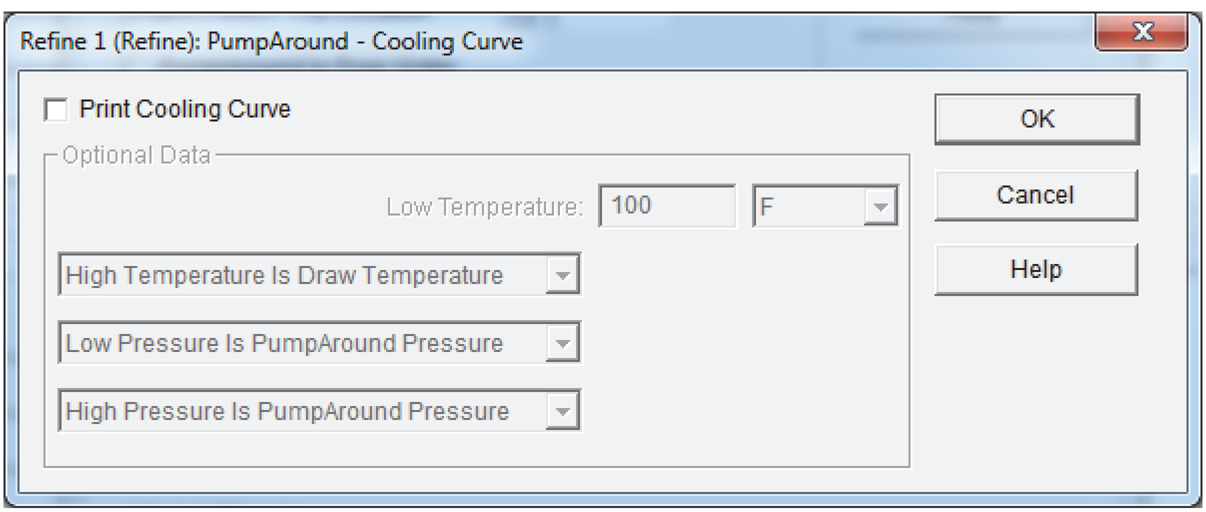


Figure 12: PumpAround – Cooling Curve Dialog (refi4.psd)

Cooling Curve

Use this dialog to enter specifications for cooling curves on pumparounds attached to the main column or for products exiting the column.

If you want to print the cooling curve for a product or pumparound, click on the 'Print Cooling Curve' box. The cooling curve temperature and pressure limit items are disabled if the cooling curve is not printed.

If you have selected the cooling curve; then you can enter the low temperature, high temperature, low pressure and high pressure for the cooling curve limits. The low temperature limit default is 100 degrees F, while the high temperature limit default is the pumparound draw temperature or the product draw temperature. The default for both pressure limits is the product pressure or pumparound pressure. You can use the default value for each item or enter a new value.

Data Item	Description
Print Cooling Curve	Check to print, clear to suppress print of cooling curve.
Low Temperature	Supply the low temperature value.
High Temperature	Select either High Temperature is Draw Temperature (the value is calculated) or Cooling Curve High Temperature (you must enter the value).
Low Pressure	Select either Low Pressure is Pumparound Pressure (the value is calculated) or Cooling Curve Low Pressure (you must enter the value).
High Pressure	Select either High Pressure is Pumparound Pressure (the value is calculated) or Cooling Curve High Pressure (you must enter the value).

Specify Feed

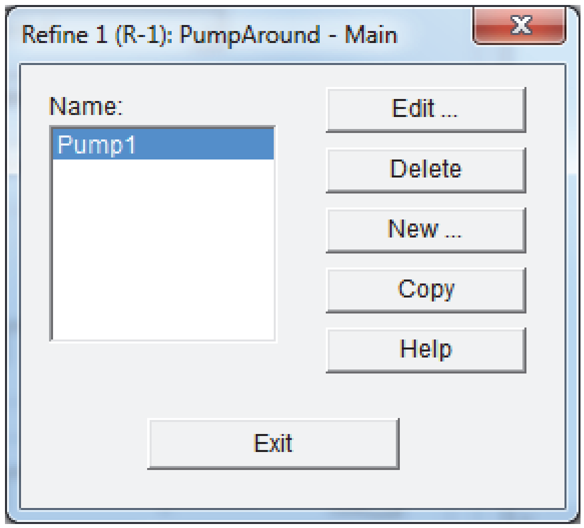


Figure 13: Feed - Main Dialog (from refi3.psd)

Use this dialog to assign specifications for the Feed of the Refine column. The dialog consists of a list box called Name and a set of buttons. Use the buttons to perform various operations on the Feed; two of the buttons display additional dialogs.

The Name list box displays Feeds for the column. DESIGN II for Windows assigns default names. You create a Feed and an entry in the list box by selecting the New dialog. If you decide not to use this Feed, delete it before leaving this dialog. To delete a Feed and its values from the list in Name, select the Feed ; then click on the Delete button.

To copy a PumpAround and its values, select the Feed ; then click on the Copy button. To change the specifications of an existing Feed, click on the Edit dialog button.

When you are finished creating Feeds for the column, or editing existing Feeds, click on the Exit button.

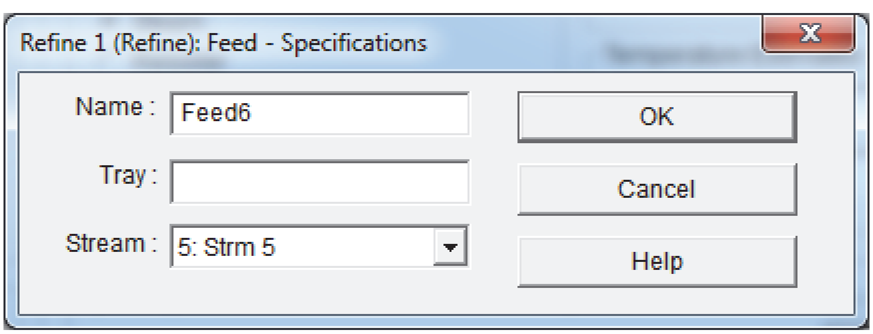


Figure 14: Feed – Main (from refi4.psd)

You can assign flowsheet streams as feeds to the distillation/Refine column and assign the feed to a column tray.

This dialog is used to enter the name, tray location, and stream identity for column feeds. The data entries are described in the table below.

Feeds can enter on any theoretical tray in the column. The top tray is always tray one and the condenser, if present, is tray zero. Any liquid in the feed will go on the feed tray. This means if the feed is two phase at the tray pressure; then the liquid portion of the feed enters the feed tray and the vapor portion automatically mixes with the liquid in the tray above. If the feed is all vapor; then all of the feed enters and mixes with the liquid in the tray above the feed tray.

Feeds are always adiabatically flashed at the tray pressure. If the feed pressure is less than the tray pressure, calculations will continue but a warning message will state that a feed pump is required.

Data Item	Description
Name	Enter the name of the feed. If no entry is made, the default name that appears here will be used for the feed name.
Tray	Enter the tray location for the feed. The top tray in the column is always tray number one.
Stream	This is a combo box that allows you to choose which of the flowsheet streams represents the feed. When you select the Stream combo box, a list of flowsheet streams will appear that you have created for the flowsheet. If you select one of these streams, its name will appear at the top of the box. Alternatively, you can enter the stream name directly in the edit portion of the combo box.

Required Specifications: Main (Pumparound with Condenser – Refine 3 / 3A symbol)

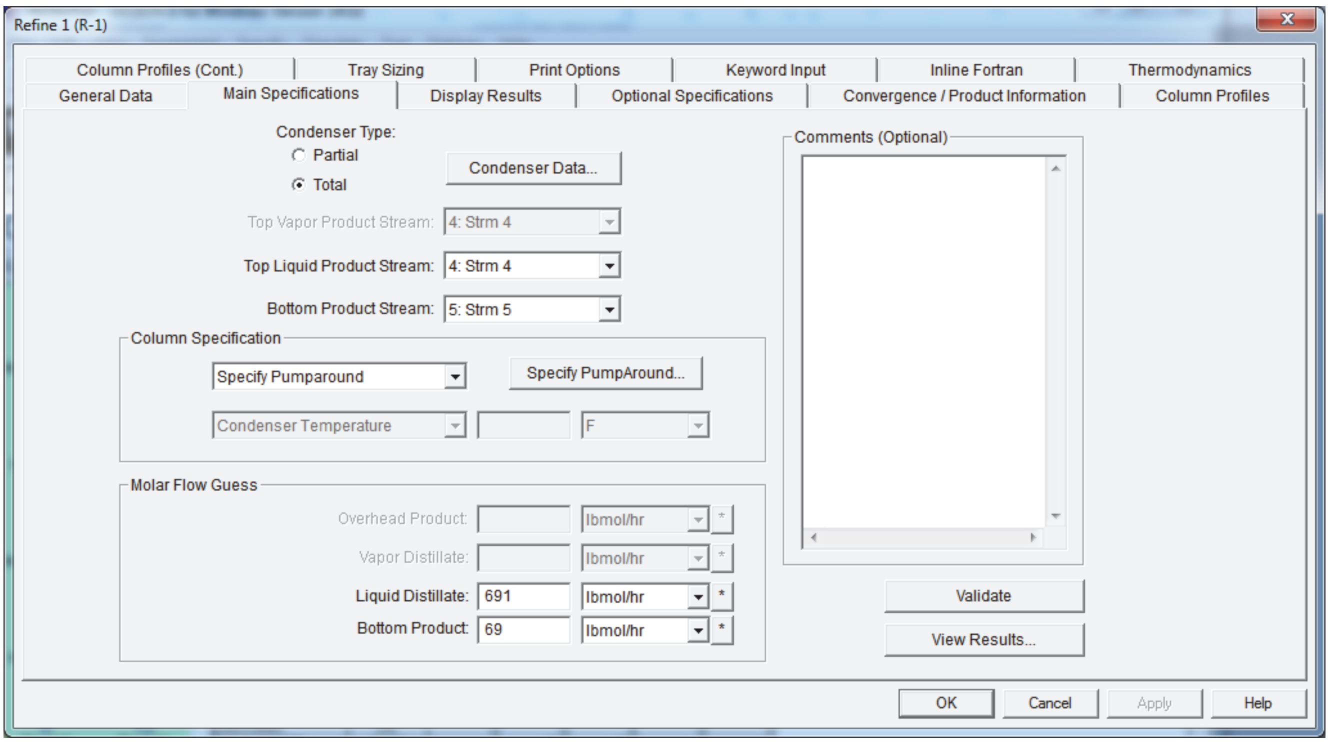


Figure 15: Main Specifications Tab (from refi4.psd)

Enter the main specifications for Refine columns using this dialog, organized into group boxes, radio buttons and cascading dialog buttons. This dialog combines elements from the Refine 1/1A and Refine 2/2A Main Specifications tabs. Refer to the documentation on completing those tabs for more details.

Optional Specifications

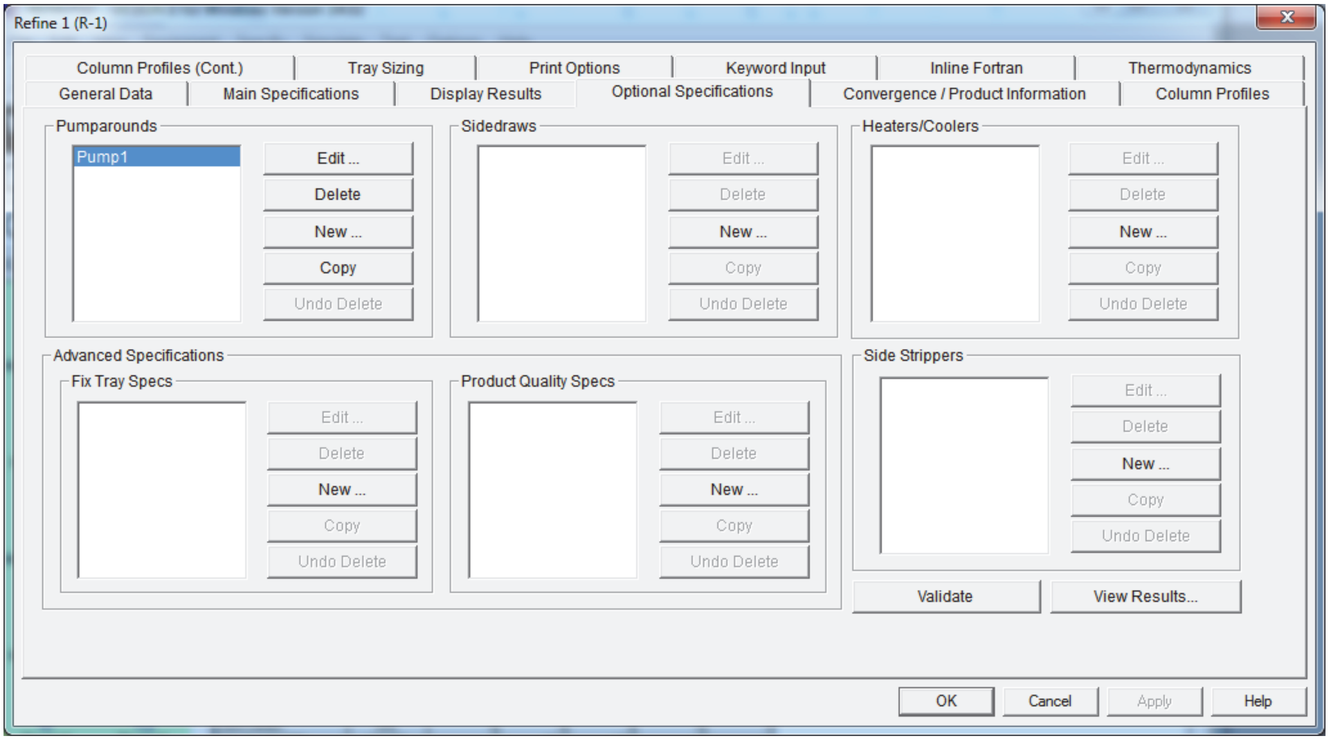


Figure 16: Optional Specifications Tab (from refi4.psd)

PumpAround Specifications

Enter the specifications for pumparounds in Refine columns using this dialog, organized into group boxes and cascading dialog buttons.

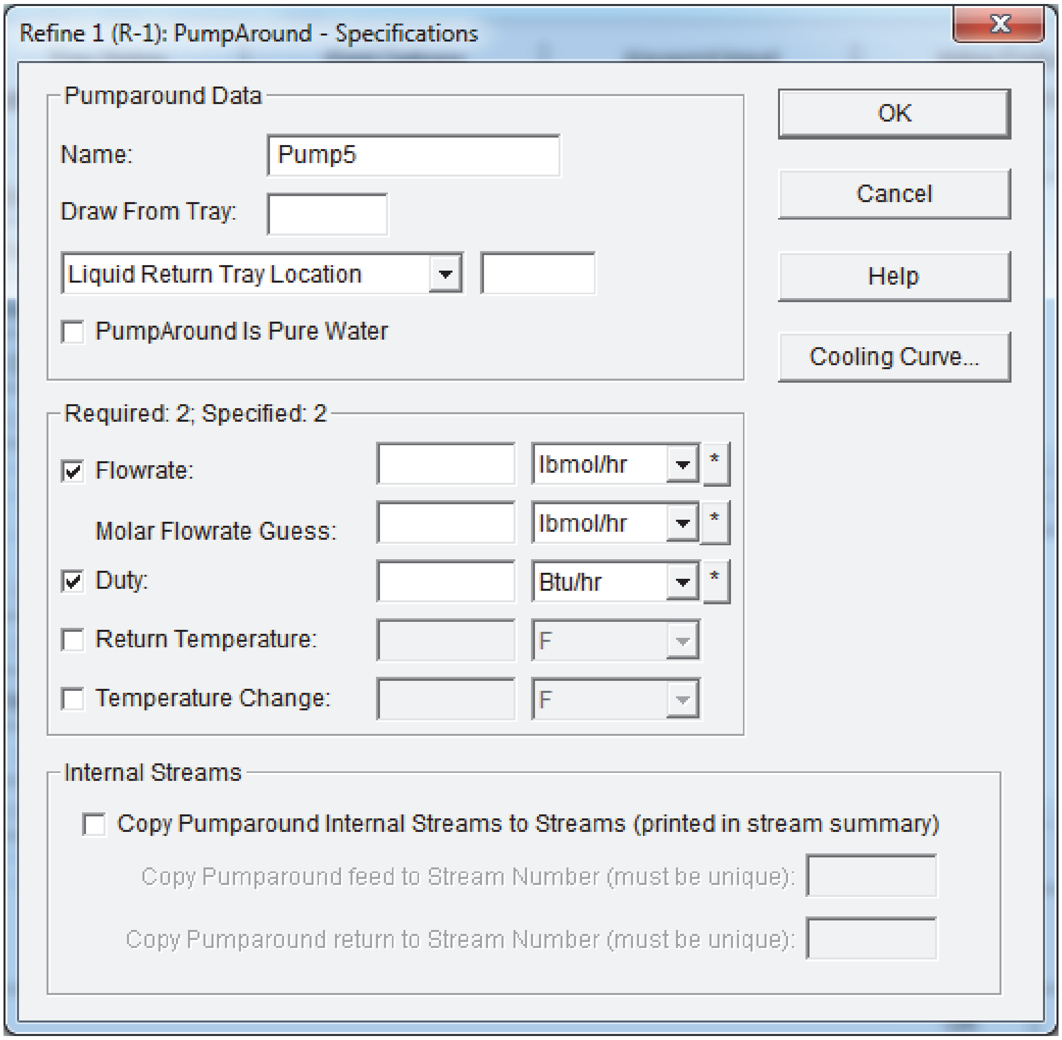


Figure 17: PumpAround – Specifications (from refi4.psd)

Use this group box to enter specifications for pumparounds attached to the Refine column. The pumparound name is automatically generated but you can change it. You must specify the pumparound draw tray location in the main column.

The Refine column allows for liquids to be pumped up (the return tray is higher than the draw tray), pumped around (the return and draw are on the same tray), and pumped down (the return tray is below the draw tray). Additionally, you are allowed to mix a pumparound with a feed to the main column.

The pumparound can return to a tray in the main column or it can be mixed with a feed to the main column. The default pumparound return is to a tray in the main column. If the pumparound return is to a feed; then the value entered for the return location is the position of the feed in top to bottom order. For example, the top feed is 1, the second feed from the top is 2, and the fourth feed from the top is 4. The return tray number or return feed number must be entered.

If the pumparound contains pure water; then check the Pumparound Is Pure Water box to indicate so. The default composition for pumparounds is a hydrocarbon stream.

NOTE: Water pumparounds require water Partial Solubility profile.

Data Item	Description
Name	View the name or change it. A name can contain up to 16 characters; however, all the product names combined should not exceed 65 characters.
Draw From Tray	Enter the draw tray for the pumparound.
Liquid Return	Select either: Liquid Return Tray Location: Enter the return tray for the pumparound. Liquid Return Feed Number: Enter the feed number with which the pumparound return mixes.
Pumparound Is Pure Water	Check the box if the pumparound is pure water. The default composition for pumparounds is a hydrocarbon stream. **Note**: Water pumparounds require a Water Partial Solubility profile.

Required/Specified

View the number of required specifications and the number already specified. You must select two of the following four specifications for the pumparound: flowrate, duty, return temperature and temperature change. We strongly urge you to use the flowrate of the pumparound liquid as one of the specifications. No heating is allowed for pumparounds.

Flowrate: Click the checkbox; then enter the flowrate of the pumparound liquid and select a unit. To enter base quantity and time units, click the *.

Molar Flowrate Guess: Enter the estimated molar flowrate and select a unit. To enter base quantity and time units, click the *. If the pumparound flowrate has been specified in non-molar dimensional units then the molar flowrate guess must be entered. If the pumparound flowrate was not specified then we recommend that the molar flowrate guess be entered.

Duty: Click the checkbox; then enter the amount of heat removed from the pumparound and select a unit; enter this value as a positive number. To enter base quantity and time units, click the *.

Return Temperature: Click the checkbox; then enter the return temperature of the pumparound and select a unit.

Temperature Change: Click the checkbox; then enter the temperature drop through the pumparound and select a unit.

Internal Streams

Copy Pumparound Internal Streams to Streams:

Select this checkbox to add internal stream data for the pumparound to a stream, so that the data will appear in the stream summary output.

Copy Pumparound feed to Stream Number (must be unique):

Enter a stream number.

Copy Pumparound return to Stream Number (must be unique):

Enter a stream number.

Cooling Curve

Click this button to enter pumparound cooling curve data.

PumpAround Cooling Curve

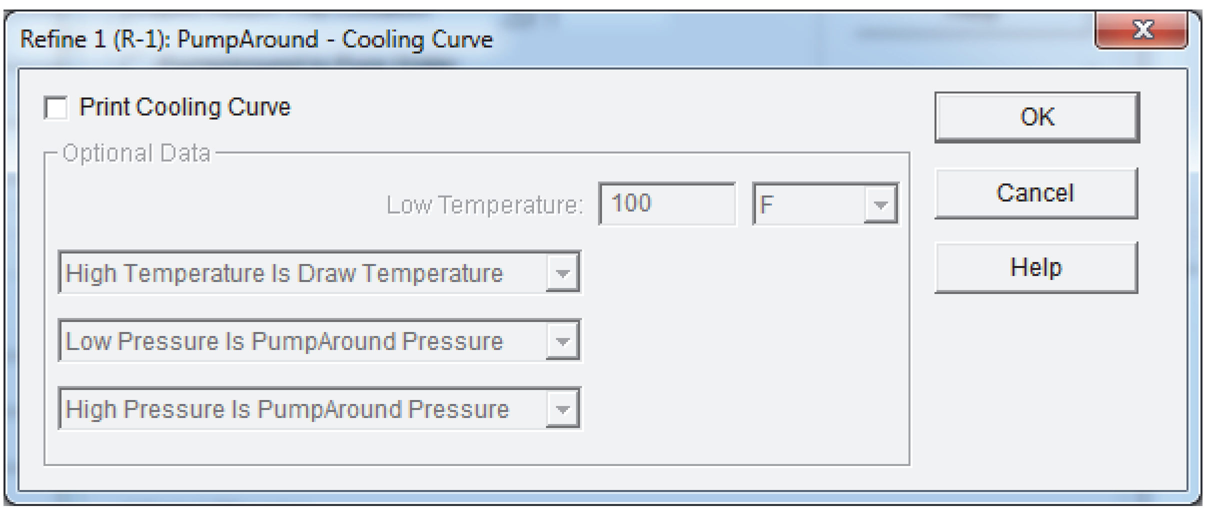


Figure 18: PumpAround – Cooling Curve Dialog (refi4.psd)

Use this dialog to enter specifications for cooling curves on pumparounds attached to the main column or for products exiting the column.

If you want to print the cooling curve for a product or pumparound, click on the 'Print Cooling Curve' box. The cooling curve temperature and pressure limit items are disabled if the cooling curve is not printed.

If you have selected the cooling curve; then you can enter the low temperature, high temperature, low pressure and high pressure for the cooling curve limits. The low temperature limit default is 100 degrees F, while the high temperature limit default is the pumparound draw temperature or the product draw temperature. The default for both pressure limits is the product pressure or pumparound pressure. You can use the default value for each item or enter a new value.

Data Item | Description

Print Cooling Curve	Check to print, clear to suppress print of cooling curve.
Low Temperature	Supply the low temperature value.
High Temperature	Select either High Temperature is Draw Temperature (the value is calculated) or Cooling Curve High Temperature (you must enter the value).
Low Pressure	Select either Low Pressure is Pumparound Pressure (the value is calculated) or Cooling Curve Low Pressure (you must enter the value).
High Pressure	Select either High Pressure is Pumparound Pressure (the value is calculated) or Cooling Curve High Pressure (you must enter the value).

Optional Specifications: Side Draws

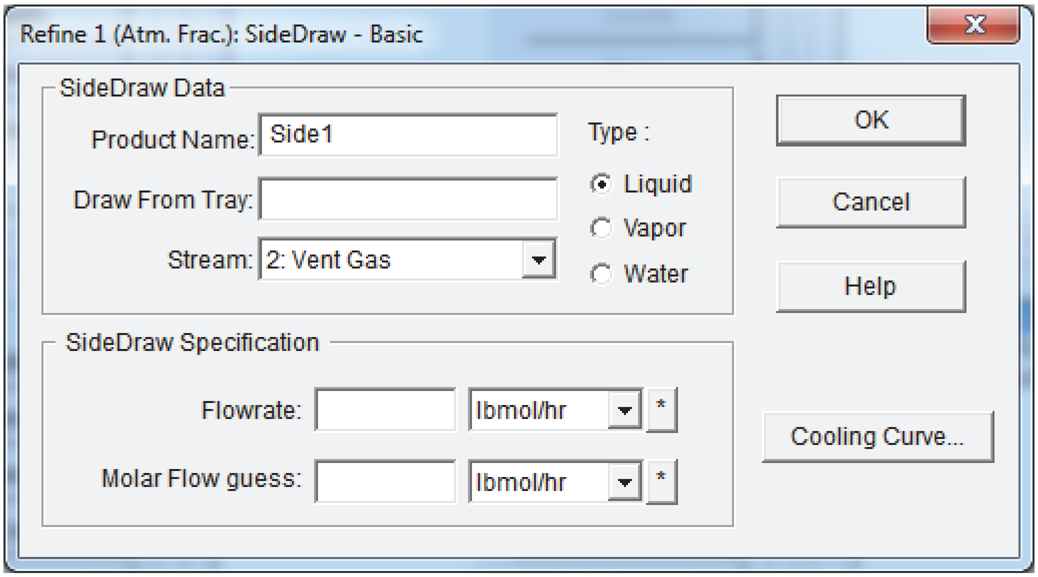


Figure 19: Side Draw – Basic (from refi5.psd)

Use this dialog to assign specifications for the sidedraws connected to the Refine column.

Enter the specifications for sidedraws attached to Refine columns using this dialog, organized into group boxes and cascading dialog buttons. You should have an entry or a selection in each of the fields on this dialog.

Refine- Sidedraw Data
Use this group box to enter data for sidedraws attached to the Refine column. The sidedraw name is automatically generated but you can change it. Sidedraw product names can contain up to 16 characters; however, all the product names combined should not exceed 65 characters. You must specify the draw tray from the main column and the flowsheet stream to which the sidedraw product is connected.

The draw tray specified must be within the range of trays on the main column; it cannot be attached to the condenser or reboiler on the main column. The types of sidedraw are liquid (default), vapor, or water. Water sidedraws require a Water Partial Solubility Profile.

Data Item	Description
Product Name	Enter the name of the product from the sidedraw.
Draw From Tray	Enter the draw tray for the sidedraw.
Stream	Select the stream on the flowsheet to which the sidedraw product is attached.
Sidedraw Type	Select the type of sidedraw: Liquid (default), Vapor, or Water.

Refine- Sidedraw Specifications
Use this group box to enter required flowrate specifications for sidedraws attached to the Refine column. The sidedraw flowrate must be entered. If you specified the sidedraw product flowrate in mass or volume (non-molar) units, we recommend that you enter a molar product flowrate guess. If you do not enter a molar guess, Refine will perform a Fenske shortcut calculation.

Data Item	Description
Flowrate	Enter the product flowrate from the sidedraw.
Molar Product Guess	Enter the product flowrate guess from the sidedraw.

Side Draw Cooling Curve

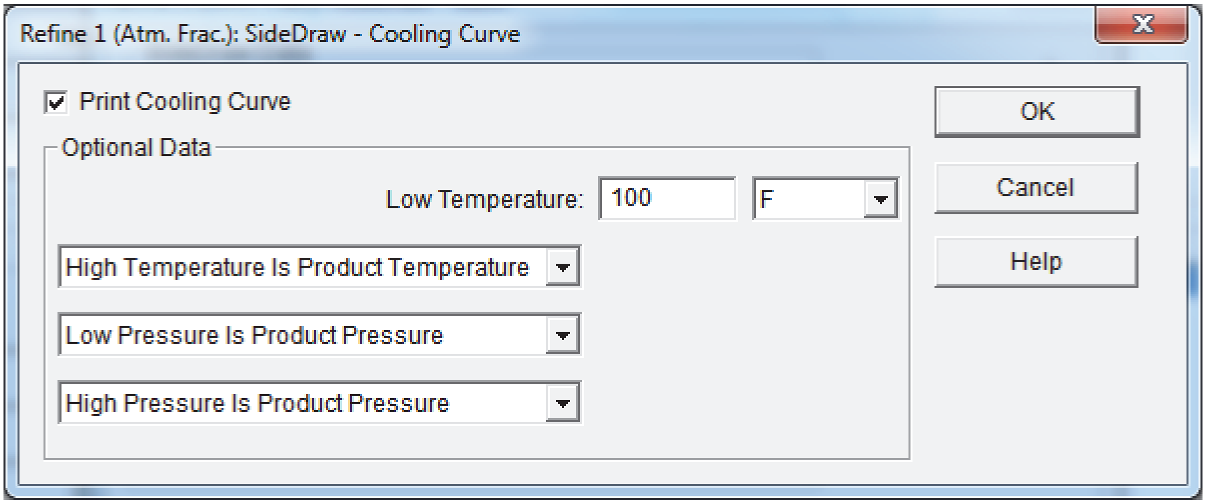


Figure 20: Side Draw – Cooling Curve Dialog (refi4.psd)

Use this dialog to enter specifications for cooling curves on pumparounds attached to the main column or for products exiting the column.

If you want to print the cooling curve for a product or pumparound, click on the 'Print Cooling Curve' box. The cooling curve temperature and pressure limit items are disabled if the cooling curve is not printed.

If you have selected the cooling curve; then you can enter the low temperature, high temperature, low pressure and high pressure for the cooling curve limits. The low temperature limit default is 100 degrees F, while the high temperature limit default is the pumparound draw temperature or the product draw temperature. The default for both pressure limits is the product pressure or pumparound pressure. You can use the default value for each item or enter a new value.

Data Item	Description
Print Cooling Curve	Check to print, clear to suppress print of cooling curve.
Low Temperature	Supply the low temperature value.
High Temperature	Select either High Temperature is Product Temperature (the value is calculated) or Cooling Curve High Temperature (you must enter the value).
Low Pressure	Select either Low Pressure is Product Pressure (the value is calculated) or Cooling Curve Low Pressure (you must enter the value).
High Pressure	Select either High Pressure is Product Pressure (the value is calculated) or Cooling Curve High Pressure (you must enter the value).

Side Strippers

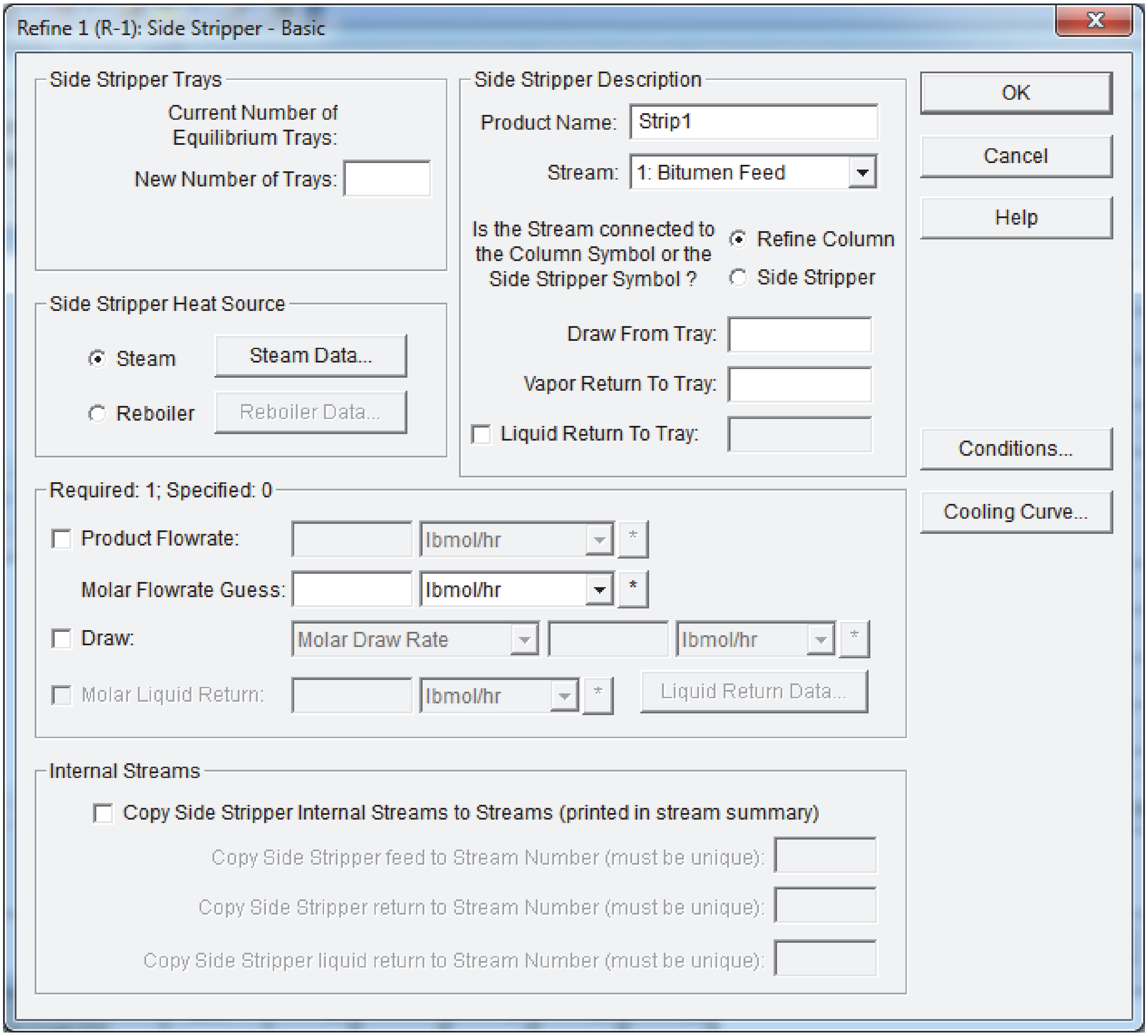


Figure 21: Sidestripper Dialog (refi4.psd)

Use this dialog to assign specifications for the side strippers connected to the Refine column.

Enter the specifications for side strippers attached to Refine columns using this dialog, organized into group boxes and cascading dialog buttons. You should have an entry or a selection in each of the fields in the Basic dialog. A heat source, steam or reboiler, must also be defined.

Side Stripper Trays

Use this group box to enter the number of trays in the column (not including the condenser and reboiler, if present).

The top tray is always tray 1. The trays are numbered from top-to-bottom. Do not count the condenser or reboiler in the number of trays. DESIGN II numbers the partial condenser as tray zero and the reboiler as one greater than the number of trays in the column. A minimum of two trays is required in the main column.

Data Item	Description
Current Number of Trays	Reports to you the number of trays you have specified for the column; if you have not entered the number of trays, the space will be blank.
New Number of Trays	Enter the number of trays for the column. If you change the number of trays in the column, you may need to make adjustments in other specifications, such as the location of sidedraws or feeds. When you change the number of trays a third edit box appears, which allows you to add or delete trays in a particular section of the column.

Side Stripper Description

Use this group box to enter specifications for side strippers attached to the Refine column. The side stripper name is automatically generated but you can change it. Side stripper product names are allowed up to 16 characters; all product names together should not exceed 65 characters. The location of the tray(s) in the main column from which the side stripper liquid feed is drawn and to which the vapor is to return must be specified.

If the side stripper has any liquid returning to the main column; then you must select the liquid return check box and specify the liquid return tray location in the main column. This will also change the specifications required on the side stripper from one to two and activate the molar liquid return specification and the liquid return data dialog.

Please note that you must indicate if the stripper product stream is attached to the main column (default) or a add-on side stripper symbol.

The draw and return trays specified must be within the range of trays in the main column. The draw or return cannot be attached to the condenser or reboiler on the main column.

Data Item	Description
Product Name	Enter the name of the product from the side stripper.
Stream	Select a stream from the drop-down list.
Is the Stream Connected to the Column Symbol or the Side Stripper Symbol	Select either Refine Column or Side Stripper.
Draw from Tray	Enter the main column draw tray for the side stripper feed. The draw tray specified must be within the range of trays in the main column. The draw cannot be attachedto the condenser or reboiler on the main column.
Vapor Return to Tray	Enter the return tray in the main column for the side stripper vapor. The return tray specified must be within the range of trays in the main column. The return cannot be attached to the condenser or reboiler on the main column.
Liquid Return to Tray	f the side stripper has any liquid returning to the main column; then you must select the liquid return check box and specify the liquid return tray location in the main column. This will also change the specifications required on the side stripper from one to two and activate the molar liquid return specification and the liquid return data dialog (under Required/Specified).

Side Stripper Heat Source

Use this group box to select the heat source for the side stripper; the heat source is either a steam feed below the bottom tray (default) or a reboiler below the bottom tray.

If the heat source is steam; then click the steam button; specify steam data using the dialog that displays. If the heat source is a reboiler; then click the reboiler button; specify reboiler data using the dialog that displays.

Data Item	Description
Steam	Select this radio button if you want a steam feed below the bottom tray for the side stripper heat source. Then click the Steam Data button.
Reboiler	Select this radio button if you want a reboiler below the bottom tray for the side stripper heat source. Then click the Reboiler Data button.

Required/Specified

Use this group box to enter required specifications for side strippers attached to the Refine column. Enter either one or two of the following three specifications for the side stripper.

For a side stripper without a liquid return, the specification choice is product flowrate or draw rate from the main column. If liquid is returned to the main column from the side stripper, you must fill in two of the three specification choices: product flowrate, draw rate from the main column, or return liquid rate. You can specify the draw rate in flow units or as a fraction of the total liquid leaving the tray. For a total draw tray, specify Draw rate fraction=1. If you specify Draw Rate on a fractional basis for one side stripper, any Draw Rates specified for the other side strippers attached to the same Refine column must also be entered on a fractional basis.

If the side stripper product flowrate was not specified; then we recommend that you enter a molar product flowrate guess. Or, if the product flowrate was specified in mass or volume (non-molar) units, an estimate of the product molar flowrate may speed convergence.

Data Item	Description
Product Flowrate	Enter the product flowrate from the side stripper.
Molar Product Flowrate Guess	Enter the product flowrate guess from the side stripper.
Draw	Enter the draw flowrate or draw fraction from the main column to the side stripper.
Molar Liquid Return	Enter the liquid return from the side stripper to the main column.
Liquid Return Data	Click this button to enter liquid return data.

Internal Streams

Copy Side Stripper Internal Streams to Streams:	Select this checkbox to add internal stream data for the side stripper to a stream, so that the data will appear in the stream summary output.
Copy Side Stripper feed to Stream Number (must be unique):	Enter a stream number.
Copy Side Stripper return to Stream Number (must be unique):	Enter a stream number.
Copy Side Stripper liquid return to Stream Number (must be unique):	Enter a stream number.
Conditions	Click this button to enter condition data for the side stripper.
Cooling Curve	Click this button to enter cooling curve data for the side stripper.

Side Stripper Steam Data

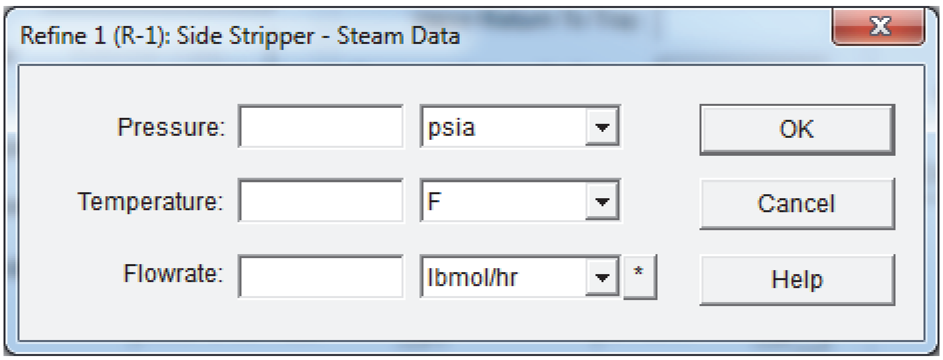


Figure 22: Side Stripper – Steam Data (from refi5.psd)

Use this dialog to enter a optional return liquid cooling specification for the side stripper. One of the following three specifications can be entered: heat removed from the return liquid, the temperature of the return liquid, or the temperature drop of the return liquid.

Use the same dimensional units for the liquid cooling specifications as those for all the side strippers.

DESIGN II will always calculate the steam feed location for you. You must make all dimensional unit choices for pressure, temperature, and flowrates consistent for the main column and each side stripper (e.g. PRE STE (ATM)= 10,10,10)

Data Item	Description
Pressure	Enter the steam feed pressure and select a unit.
Temperature	Enter the steam feed temperature.
Flowrate	Enter the steam feed flowrate.

Side Stripper Reboiler Data

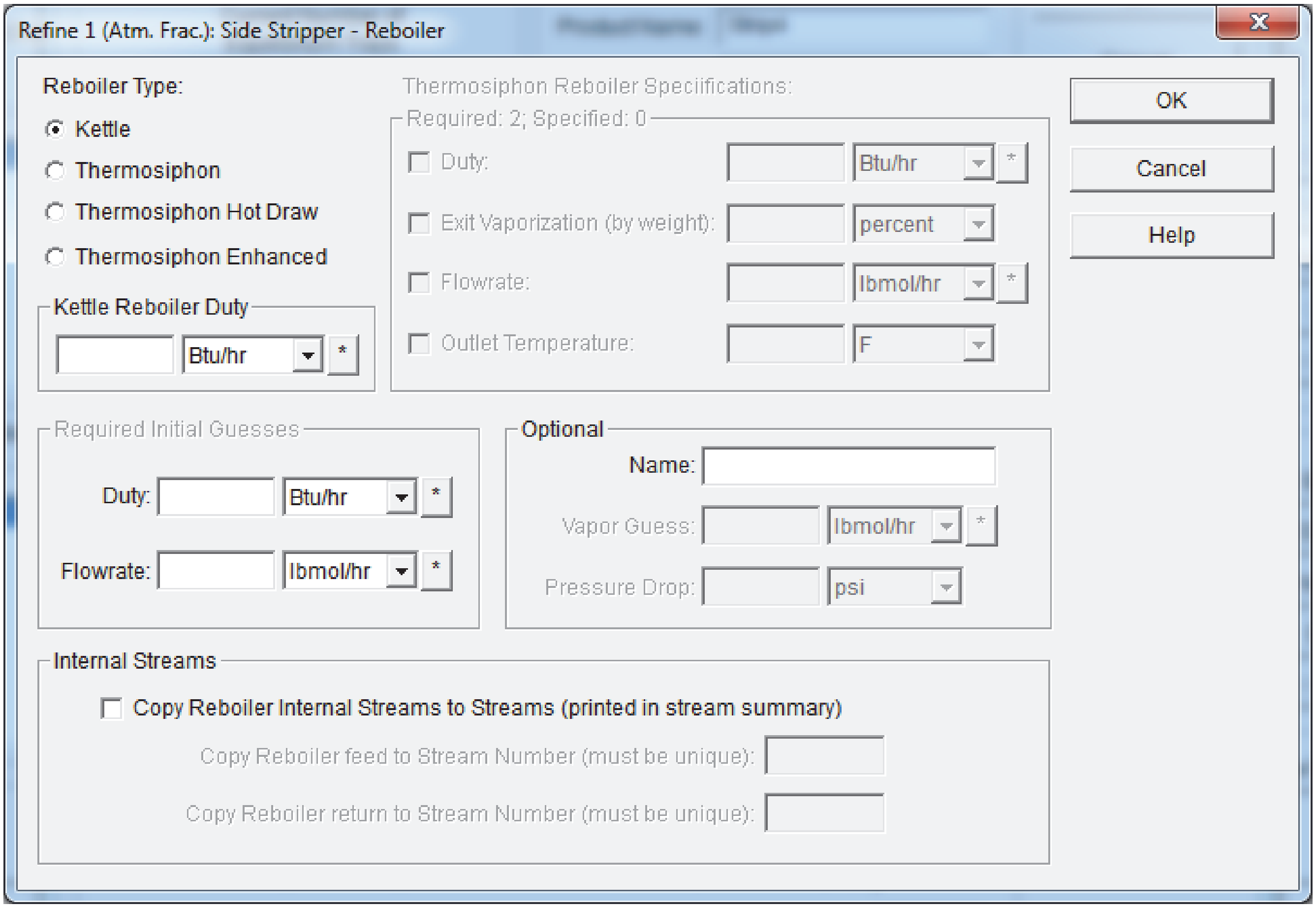


Figure 23: Side Stripper – Reboiler (from refi5.psd)

Use this dialog to enter data describing a reboiler for the main column or a side stripper. You may select one of four types of reboilers: kettle, thermosiphon, thermosiphon hot draw, or thermosiphon enhanced.

The default reboiler type is kettle; the enhanced thermosiphon reboiler includes the vaporization effects of mixing the sump liquid with the reboiler return liquid.

If the reboiler type is a kettle, you must enter a duty.

Thermosiphon reboilers require two specifications from the list below:

- Duty
- Exit vaporization percent
- Flowrate
- Outlet Temperature

You must enter a guess for reboiler duty and/or flowrate if you did not choose them as specifications. You may enter guesses for the vapor flow from the reboiler (on a molar basis) and for the pressure drop (rise) between the bottom tray and the reboiler outlet. If you do not enter a guess for the pressure drop, the value you entered for Bottom Tray Pressure will be used.

You may specify a name for each reboiler, regardless of reboiler types.

Data Item	Description
Required Initial Guess	
Duty	Enter the reboiler duty and select a unit.
Flowrate	Enter the reboiler flowrate and select a unit.
Optional	
Name	View the name or change it. A name can contain up to 16 characters; however, all the product names combined should not exceed 65 characters.
Vapor Guess	Enter a guess for the vapor flow from the reboiler (on a molar basis).
Pressure Drop	Enter a guess for the pressure drop (rise) between the bottom tray and the reboiler outlet.

Internal Streams

Copy Reboiler Internal Streams to Streams	Select this checkbox to add internal stream data for the reboiler to a stream, so that the data will appear in the stream summary output. Copy Reboiler feed to Stream Number (must be unique): Enter a stream number. Copy Reboiler return to Stream Number (must be unique): Enter a stream number.

Side Stripper Liquid Return Data

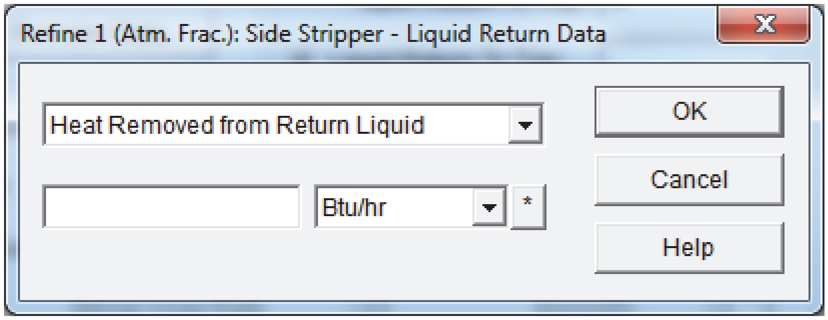


Figure 24: Side Stripper – Liquid Return Data (from refi5.psd)

Liquid Return Data

Use this dialog to enter an optional return liquid cooling specification for the side stripper. One of the following three specifications can be entered: heat removed from the return liquid, the temperature of the return liquid, or the temperature drop of the return liquid.

Use the same dimensional units for the liquid cooling specifications as those for all the side strippers.

Data Item	Description
Heat Removed from Return Liquid	Enter the amount of heat removed from the liquid which is returned from the strippers to the main column.
Temperature of the Return Liquid	Enter the temperature of the liquid returning to the main column.
Temperature Drop of Return Liquid	Enter the temperature drop of the liquid returning to the main column.

Side Stripper Condition Data

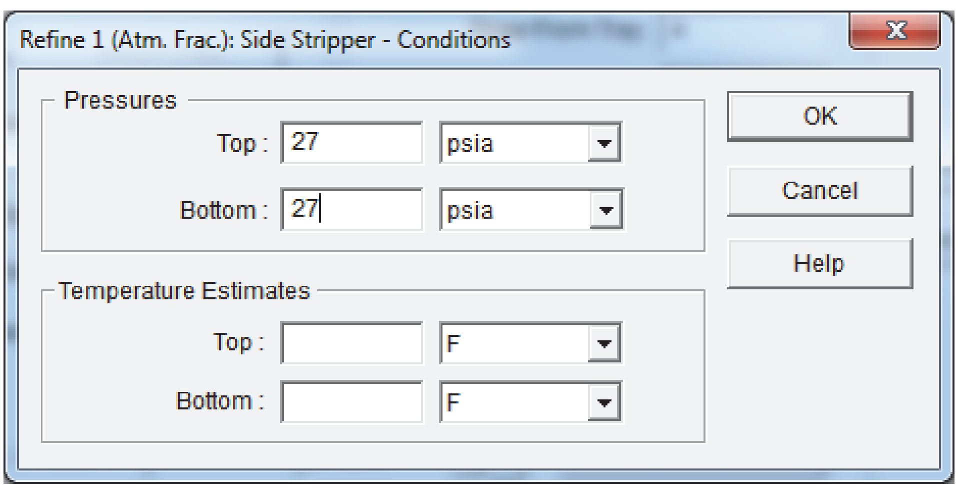


Figure 25: Side Stripper – Conditions (refi5.psd)

Use this dialog to enter pressure and temperature specifications for the side stripper. Entering the pressures and temperature for the side stripper is optional. A linear profile is generated for the trays in the side stripper from these specifications. If you have a nonlinear profile; then use the Profiles dialogs under Optional Specs.

Use the same dimensional units for the pressure and temperature values as were used for the main column and other side strippers.

Data Item	Description

Top Pressure	Enter the top tray pressure. If you have a nonlinear pressure profile, the complete Pressure Profile (a pressure for every tray) can be entered (see the Profiles/Pressures dialog).
Bottom Pressure	Enter the bottom tray pressure for the side stripper.
Top Temperature Estimate	Enter an estimate for the top tray temperature in the side stripper.
Bottom Temperature Estimate	Enter an estimate for the bottom tray temperature in the side stripper.

Side Stripper Cooling Curve

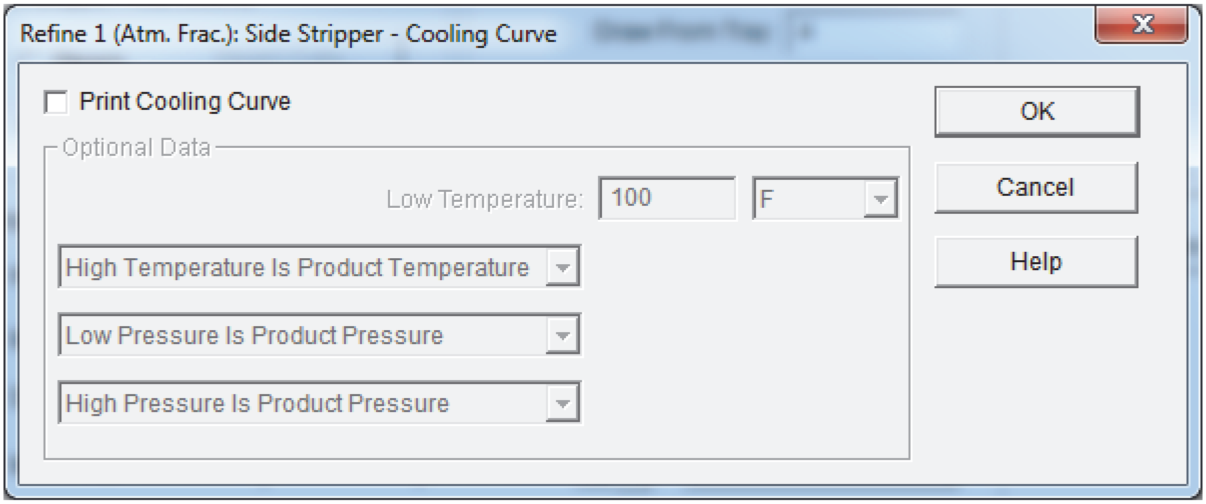


Figure 26: Side Stripper – Cooling Curve (from refi5.psd)

Use this dialog to enter pressure and temperature specifications for the side stripper. Entering the pressures and temperature for the side stripper is optional. A linear profile is generated for the trays in the side stripper from these specifications. If you have a nonlinear profile; then use the Column Profiles tab.

Use the same dimensional units for the pressure and temperature values as were used for the main column and other side strippers.

Data Item	Description
Print Cooling Curve	Check to print, clear to suppress print of cooling curve.
Low Temperature	Supply the low temperature value.
High Temperature	Select either High Temperature is Product Temperature (the value is calculated) or Cooling Curve High Temperature (you must enter the value).
Low Pressure	Select either Low Pressure is Product Pressure (the value is calculated) or Cooling Curve Low Pressure (you must enter the value).
High Pressure	Select either High Pressure is Product Pressure (the value is calculated) or Cooling Curve High Pressure (you must enter the value).

Heaters/Coolers

You use this dialog to create heaters and coolers for the Refine column, to indicate what tray on which the heater/cooler occurs, and to set the duty of the heater/cooler.

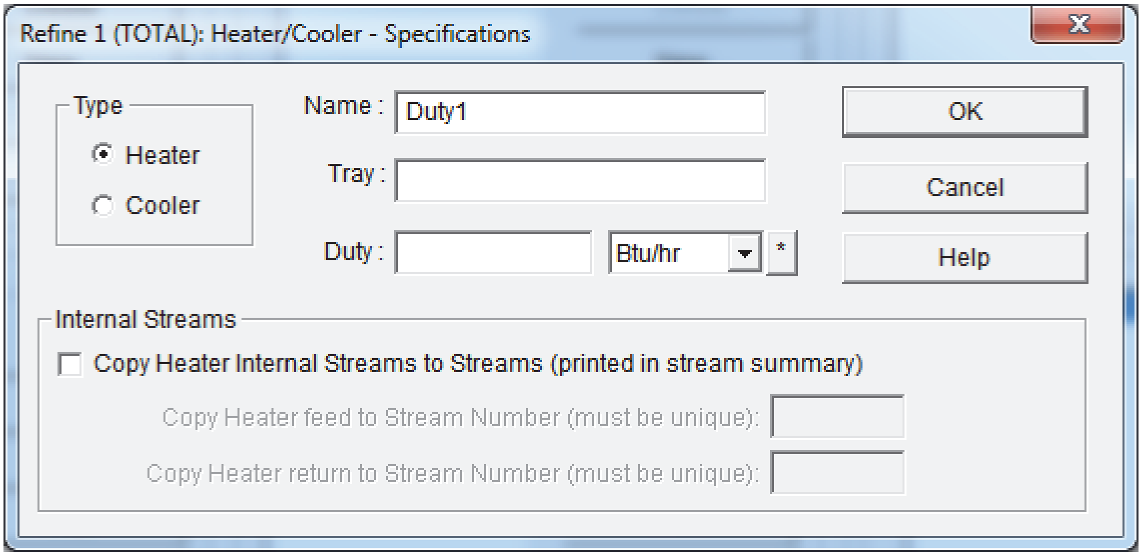


Figure 27: Heater/Cooler – Specifications (from refi1.psd)

This dialog organizes the heater/cooler specifications into radio buttons and edit boxes. These are explained in the table below.

The duty that you enter is applied directly to the tray that you specify. Liquid and vapor on the tray are mixed and the duty is added or subtracted. Finally, the temperature corresponding to the new enthalpy is calculated along with the amount of vapor and liquid.

Data Item	Description
Heater	Select this radio button to set up a heater for a column tray.
Cooler	Select this radio button to set up a cooler for a column tray.
Name	Enter the name for the heater/cooler. If the default name in the edit box is acceptable, no action is required.
Tray	Enter the tray number for the location of the heater/cooler.
Duty	Enter the duty for the heater/cooler. Always enter a positive number since the radio button determines whether or not the duty will be used for heating or cooling.
Internal Streams	
Copy Heater Internal Streams to Streams	Select this checkbox to add internal stream data for the heater to a stream, so that the data will appear in the stream summary output. Copy Heater feed to Stream Number (must be unique): Enter a stream number. Copy Heater return to Stream Number (must be unique): Enter a stream number.

Advanced Specifications: Fixed Tray Specs

Use this dialog to assign fix tray specifications for the Refine column.

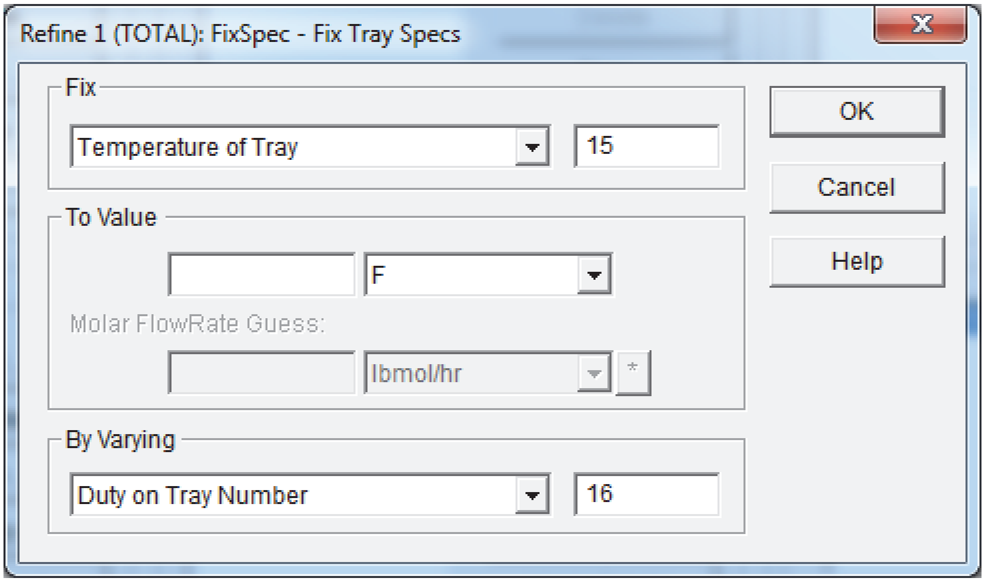


Figure 28: FixSpec – Fix Tray Specs Dialog (from refi1.psd)

Refine- Fix Tray Specs
Enter the specifications for fix specifications for trays in the Refine columns using this dialog, organized into group boxes and cascading dialog buttons. You should have an entry or a selection in each of the fields on this dialog.

Group Boxes/Basic
- Fix Tray Specs Fix
- Fix Tray Specs To Value
- Fix Tray Specs By Varying

Refine- Fix Tray Specs Fix
This set of commands allows you to fix (or set) an internal tray value which would normally be determined by column calculations by varying a condition which would normally be set (pumparound duty, feed furnace, product rate, reboiler duty, etc.)

Use this group box to enter data for the tray being fixed in the Refine column. You must enter the number of the tray being fixed and select a fix specification. The tray that you fix can be on the main column or on a side stripper. If the tray is on a side stripper; then the tray location is calculated by adding the number of trays in the main column with the number of trays above the tray in the side stripper. The side strippers are ordered from the top to the bottom using the side stripper draw tray as the reference. Add one (1) tray for each reboiler in the main column and reboiler in the side strippers between the main column and the fixed tray.

The four fix specifications are: Temperature of Tray, Liquid Flowrate from Tray, Net Liquid Flowrate from Tray, and Vapor Flowrate from Tray. The default fix specification is temperature of the fixed tray.

The most common uses of the fix commands are:
- set top tray temperature by varying the top pumparound duty,
- set flash zone temperature by varying a product draw,
- set overflash by varying a feed furnace,
- set net liquid traffic leaving a tray with both pumparound and side stripper draws by varying a pumparound duty, and
- set reflux flow by varying the main column reboiler.

For example, if you want to fix the reflux from the condenser on the main column and there is a condenser on the main column; then specify the fix tray as 0 (zero). Caution: If you want to fix the temperature of the condenser, use the temperature condenser specification on the Main Specifications dialog; do not fix the temperature of the condenser using the Fix Tray Specs.

At least one tray on the column must remain free of fixes (the temperature of condenser specification counts as a tray fix). You can only select one fix specification per tray. Product flowrates cannot be set using Fix commands; use the product flowrate specification on the side draw dialog, side stripper specification dialog, or main specification dialog.

If the main column reboiler is a thermosiphon reboiler, do not attempt to fix the bottom tray liquid and the reboiler flowrate. If you are varying the duty on the bottom stage of the column; then the reboiler duty has to be a column specification.

Data Item	Description
Fix Tray Specification	Select a tray specification to fix.
Tray to Fix	Enter a tray to be fixed.

Refine- Fix Tray Specs To Value

Use this group box to enter the value data for the fixed tray specification in the Refine column. Depending on the specification selected in the fix group box (temperature, liquid flowrate, net liquid flowrate or vapor flowrate), the target value is entered accordingly.

For fix specifications for liquid, net liquid, and vapor which are entered on a non-molar basis, you must enter a molar guess.

Data Item	Description
To Value	Enter the fixed tray temperature, liquid flowrate, net liquid flowrate or vapor flowrate
Molar FlowRate Guess	Enter the flowrate guess if the fix tray value is a flowrate and has non-molar dimensional units

Refine- Fix Tray Specs By Varying

Use this group box to indicate which item is being varied to meet the fix specification in the Refine column: tray duty, feed duty, or product rate. The number of the tray, feed or product being varied must be entered and a vary specification selected. The tray being varied can be on the main column or on a side stripper. If the tray is on a side stripper; then the tray location is calculated by adding the number of trays in the main column with the number of trays above the tray in the side stripper. The side strippers are ordered from the top to the bottom, using the side stripper draw tray as the reference. Add one (1) tray for each reboiler in the main column and reboiler in the side strippers between the main column and the fixed tray.

For example, let's say that you want to vary the third side stripper reboiler duty to fix the vapor rate returning from the third side stripper. The main column has 30 trays and a reboiler. The first side stripper from the top has 4 trays and a reboiler. The second side stripper from the top has 5 trays and no reboiler. The third side stripper has 8 trays and a reboiler. The tray to be fixed is 30 (main column trays) + 1 (main column reboiler) + 4 (first side stripper trays) + 1 (first side stripper reboiler) + 5 (second side stripper trays) + 8 (third side stripper trays) + 1 (third side stripper reboiler = 50.

The three vary specifications are: Duty on a Tray (default), Heater on a Feed, and Flowrate of a Product. The feeds are numbered from the top of the column to the bottom of the column depending on the inlet tray; the products are numbered from the top of the column to the bottom of the column depending on the draw tray from the main column.

If you are varying duty on a tray; then the varied tray must be at or below the location of the fixed tray. Only one variable is allowed per tray. For example, you cannot vary both the feed heater for a feed into tray 19 and the tray duty for tray 19 in order to satisfy two separate fixes.

Data Item	Description
By Varying	Select a specification to vary.
Tray, Feed or Product to Vary	Enter the positional number of the tray, feed or product to be varied.

Advanced Specifications: Product Quality

Enter the product quality specifications for Refine columns using this dialog.

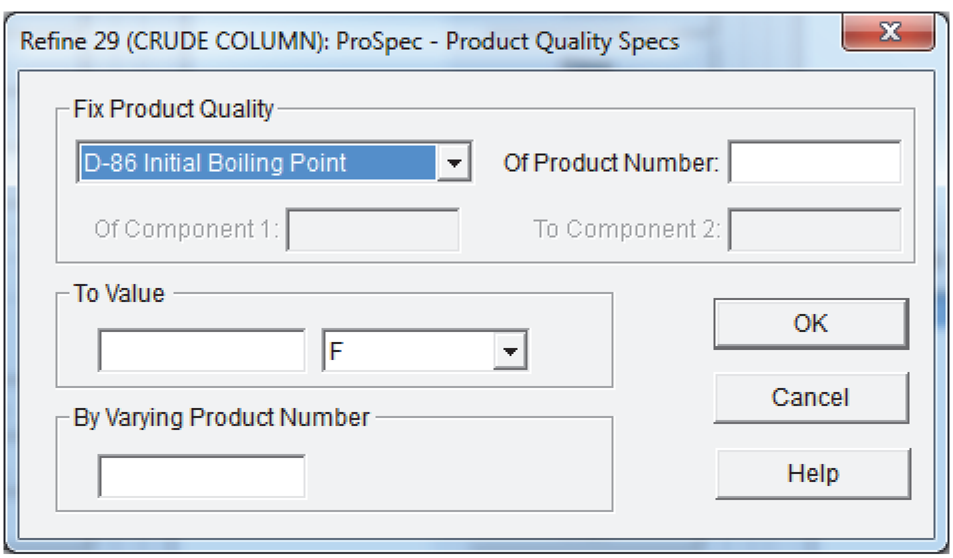


Figure 29: ProSpec – Product Quality Specs Dialog (from refinery.psd)

The product quality specification command should not be used until the Refine column has been converged. No more than 10 volume percent of any product should have to be shifted to achieve the quality spec.

More than a 30 volume percent shift of one or more products can lead to convergence difficulties. Be sure to allow for additional heat in the column if much additional material will be drawn from trays located above the main feed tray.

Enter the specifications for product qualities in a Refine column (and its stripper columns, if any) using this dialog, organized into group boxes and radio buttons. You should have an entry or a selection in each of the fields on this dialog.

Refine- Product Quality Specs Fix Specification Of Product Number
Use this group box to enter data for the product quality being fixed in the Refine column. You must enter the product number and select a product quality specification. The products are ordered from the top to the bottom of the Refine column based on product draw tray location. Products can be vapor from the top of the column, vapor from the top of the condenser, liquid from the condenser, liquid/vapor from a side draw, liquid from a side stripper, or liquid from the bottom of the column.

For example, if the column has a partial condenser and a side stripper; then the column has four products. The column's products are top vapor, top liquid, side stripper 1, and bottom liquid. If you want to specify an ASTM initial boiling point on the side stripper; then the product number is 3. If you want to specify a flash point on the top liquid product then that product number is 2.

At least one product on the column must remain free of quality fixes. There can only be one quality fix per product in the column. The product quality specification choices are:

1. ASTM (D-86) initial boiling point *
2. ASTM (D-86) 5% point
3. ASTM (D-86) 95% point
4. ASTM (D-86) end point *
5. ASTM (D-86) gap - 5% point of the next heavier product minus the 95% point of specified product
6. TBP initial boiling point *
7. TBP 5% point
8. TBP 95% point
9. TBP end point *
10. Mole fraction of the nth component in the component list
11. Total mole fraction of the n’th to the m’th components in the components list
12. Average molecular weight
13. Average gravity
14. Flash point
15. Mass or volume flow rate

* The default value for initial boiling point is 2%. The default for the end point is 98%. These values can be changed using the General section commands PROIBP = and PROEP = under 'Specify- Keyword Input'.

If you select product quality specification 10 (mole fraction of component); then you must enter the position of the desired component from the list of components that you have selected. The components are selected in the 'Specify- General-Components' dialog.

For example, let's say you have selected five components to use: water, Ethane, Propane, I-Butane and N-Butane. You want to set the purity of the Propane component to be 6.5%. The position of Propane in the component list is 3. If you select product quality specification 11 (total mole fraction of components); then you must enter the start and the end of the range of the component numbers to use. The component ending the range of components for the total mole fraction specification must have a higher boiling point than the component beginning range.

You cannot set the initial boiling point or the 5% point on the lightest product in the column. Correspondingly, you cannot set the 95% point or endpoint on the heaviest product in the column which is always the bottom liquid product.

Data Item	Description
Product Quality Specification	Select a product quality specification.
Product Number to Fix	Enter a product number to be fixed.
Of Component 1	Enter the position of the desired component in the component list. This item will be specified for both Mole Fraction and Total Mole Fraction.
To Component 2	Enter the position from the components lists for the component which ends the range to be used for the total mole fraction specification.

Refine- Product Quality Specs To Value
Use this group box to enter the product quality specification value being fixed in the Refine column. Depending on the specification type selected in the fix group box (temperature, flowrate, gravity, purity), the target value is entered accordingly.

Data Item	Description
To Value	Enter the value of the product quality specification.

Refine- Product Quality Specs By Varying Product Number

Use this group box to enter the number of the product to vary in the Refine column. However, any one product cannot be varied to meet two or more products specs. The products are ordered in the top to the bottom draw order from the Refine column. Products can be vapor from the top of the column, vapor from the top of the condenser, liquid from the condenser, liquid/vapor from a side draw, liquid from a side stripper, or liquid from the bottom of the column.

For example, if the column has a partial condenser and a side stripper; then the column has four products. The column's products are top vapor, top liquid, side stripper 1, and bottom liquid. If you want to set an ASTM initial boiling point on the side stripper by varying the bottom product; then the product number is 4. If you want to set a flash point on the top liquid product by varying the side stripper product; then that product number is 3.

The draw location of the product being varied must be at or lower than the product whose quality is being fixed in the column. Normally, the product rate to be varied to meet the quality specification is dependent on your choice of main specifications. If you have selected Top Product (Vapor, Liquid, or Overhead), you should specify that the same product flowrate is to be adjusted. For example, PRO SPEC (C) 1 = ASTM 95, 275, 3, 3.

If you have chosen Bottom Product Flowrate, you would vary the next heavier product in the column. For example, PRO SPEC 2 = TBP EP, 350, 4, 5.

If your quality specification is applied to the Bottom Product from the column, the Bottom Product flowrate must be varied, regardless of main specification choices.

Data Item	Description
By Varying Product Number	Enter the number of the product to be varied.

Convergence/Product Information

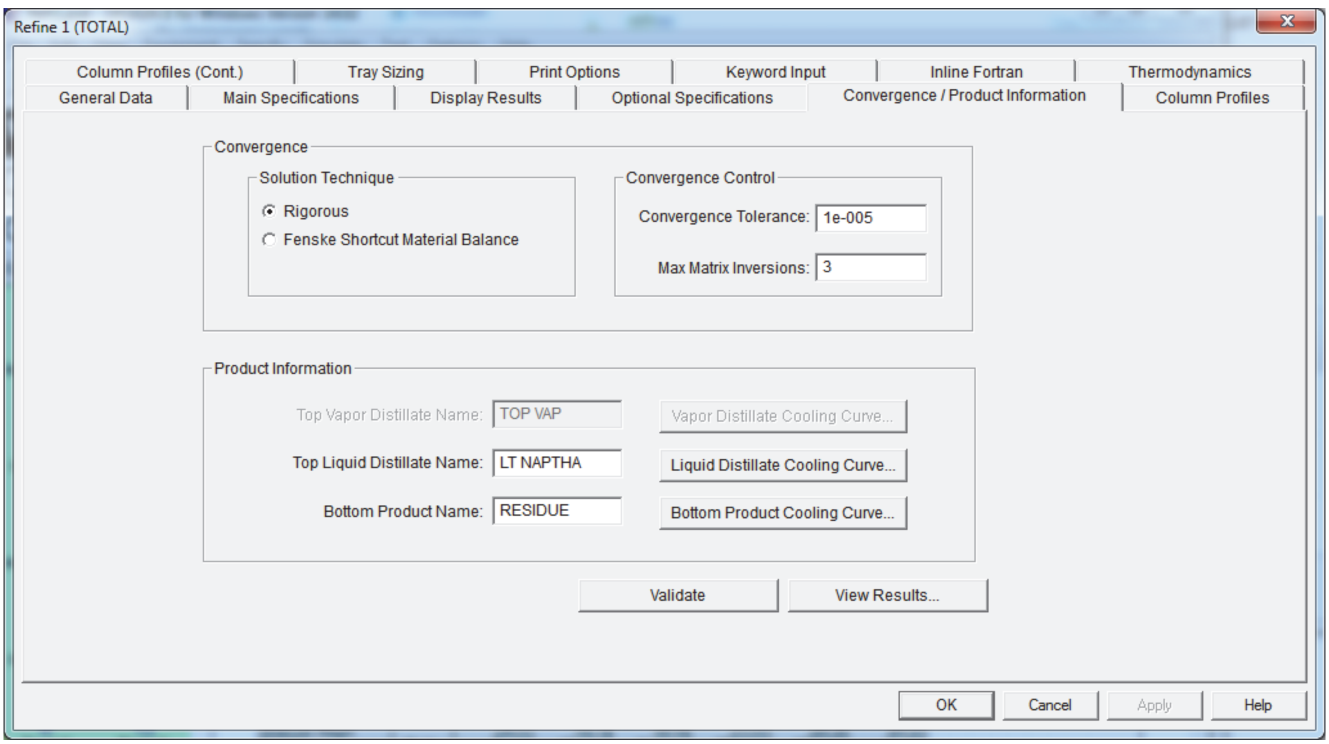


Figure 30: Convergence/Product Information Tab (from refi1.psd)

Use this dialog to enter product information and data for converging the Refine column. You should make an entry or a selection in each of the fields on this dialog.

The Refine column can use either the rigorous (tray-by-tray) or the Fenske shortcut material balance convergence technique. If you are unsure about the products splits, it is a good idea to make an initial run with the Fenske calculation command. The results allow you to review the product slate quickly and make any adjustments to Product or Pumparound flowrate guesses before your rigorous run. The Fenske command cannot be used with a condenser duty and reflux ratio specification or a condenser temperature and reflux ratio specification. Your final Refine simulation should use the rigorous calculation option.

The Heat/Material Balance Tolerance is the maximum accepted norm for the column. The norm is the error in the tray heat and material balance on a fractional basis squared. The default is 0.1E-4, or about 0.3% average error.

The maximum value for matrix inversions is 6. Maximum matrix inversions is normally set to 3; the minimum number is 1.

Data Item	Description

Convergence Technique	Select either the Rigorous button or the Fenske Shortcut button.
Heat/Material Balance Tolerance	Enter the calculation tolerance for the column.
Max Matrix Inversions	Enter the maximum number of matrix inversions for the column.

Product Information

You must enter the product names; you can also enter cooling curve data for the top and bottom products by clicking the appropriate product button and entering data in that dialog.

The three columns represented in this dialog are the top vapor product, top liquid product, and the bottom product. To edit the other column products, use the sidedraw and side stripper dialogs. The bottom product will always be present in the column. The top liquid product will be present in the column if the column has a condenser. The top vapor product will be present in the column if there is no condenser or if there is a partial condenser. The product name edit fields are dimmed if that product is not present in the column.

The default product names are 'TOP VAP', 'TOP LIQ' and 'BOTTOM'; you can change the name if desired. The product name may be up to 16 characters long. It must not contain the phrase 'FLA'. We recommend that the product name length be kept short as all product names are allowed a total of 65 characters.

Data Item	Description
Top Vapor Distillate Name	Enter the name for the column top vapor product.
Top Liquid Distillate Name	Enter the name for the column top liquid product.
Bottom Product Name	Enter the name for the column bottom product.

Vapor Distillate Cooling Curve, Liquid Distillate Cooling Curve, Bottom Product Cooling Curve

Use these dialogs to enter cooling curve data for vapor distillate, liquid distillate, and bottom product.

Data Item	Description
Print Cooling Curve	Check to print, clear to suppress print of cooling curve.
Low Temperature	Supply the low temperature value.
High Temperature	Select either High Temperature is Product Temperature (the value is calculated) or Cooling Curve High Temperature (you must enter the value).
Low Pressure	Select either Low Pressure is Product Pressure (the value is calculated) or Cooling Curve Low Pressure (you must enter the value).
High Pressure	Select either High Pressure is Product Pressure (the value is calculated) or Cooling Curve High Pressure (you must enter the value).

Column Profiles

Use this tab to enter data for the column profiles in the Refine column; the column profiles are used for both the main column and any side stripper columns. You can specify Pressure, Temperature, Vapor Rate, or Water Partial Solubility for the column profile here, depending on which menu item was selected. You should make an entry or a selection in each of the active fields on this dialog.

The temperature, vapor rate, and water partial solubility profiles are only used for the initial conditions of the Refine column. The values may be varied throughout the simulation calculations of the Refine column. The pressure profile is used to set conditions of the Refine columns. The pressure profile will not be adjusted by the simulation calculations for the Refine column.

Following is specific information about Pressure, Temperature, Vapor Rate, and Water Solubility.

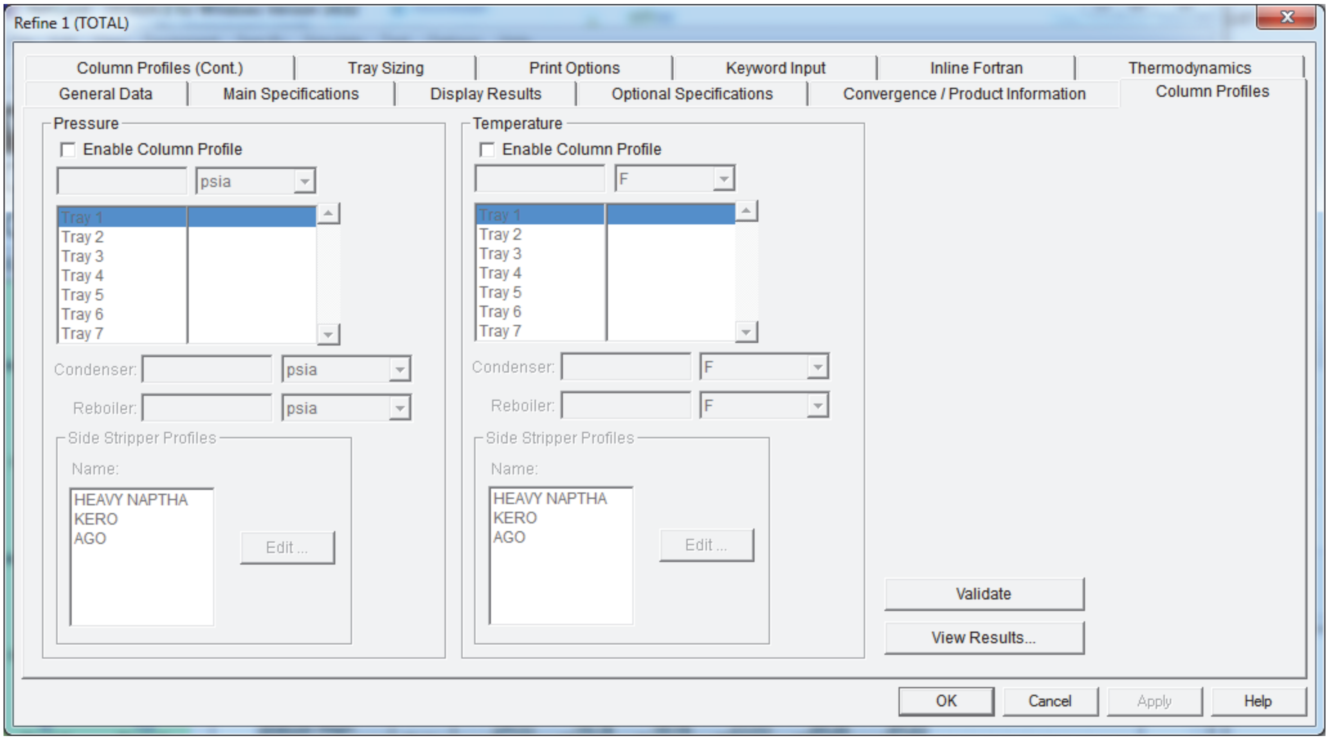


Figure 31a: Column Profiles Tab (from refi1.psd)

Pressure

The simulator will automatically set up linear pressure profiles for the main column and any side strippers based on the values you entered for Top Pressure and Bottom Pressure in the General Data tab. As column pressure can have a significant effect on vaporization for low pressure columns, you should enter a Pressure profile if your main column profile is non-linear. The values that you enter set the tray pressures; they will not be adjusted by the calculations for the Refine column.

Any zero values entered for the main column or side stripper will be linearly interpolated by the simulator, except for the first and last values. The interpolation will start at the second stage of the main column or side stripper and end at the next to last stage of the main column or side stripper.

If the values for the first and/or last stages of the pressure profile are zero; then the initial conditions from the basic specifications dialog will be substituted for these values. For best results however, the first and last stages of the main column and each side stripper should be specified, as a minimum. Otherwise, the initial conditions in the simulator will be set to zero for that property of the main column or side stripper.

Temperature

Refine automatically sets up a linear temperature profile for the main column using your initial estimates for Top and Bottom Tray Temperatures in the General Data tab. Side stripper top and bottom temperatures are based on the draw tray temperature generated by the initial temperature profile. You can adjust the side stripper column temperature estimates by entering values in the Temperature group box in the Side Strippers- Conditions dialog or by entering a temperature profile here. Heat removals/additions and feeds are common causes of non-linear temperature profiles.

Any zero values entered for the main column or side stripper will be linearly interpolated by the simulator, except for the first and last values. The interpolation will start at the second stage of the main column or side stripper and end at the next to last stage of the main column or side stripper.

If the values for the first and/or last stages of the temperature profile for the main column are zero; then the initial conditions from the basic specifications dialog will be substituted for these values. For best results however, the first and last stages of the main column and each side stripper should be specified, as a minimum. Otherwise, the initial conditions in the simulator will be set to zero for that property of the main column or side stripper.

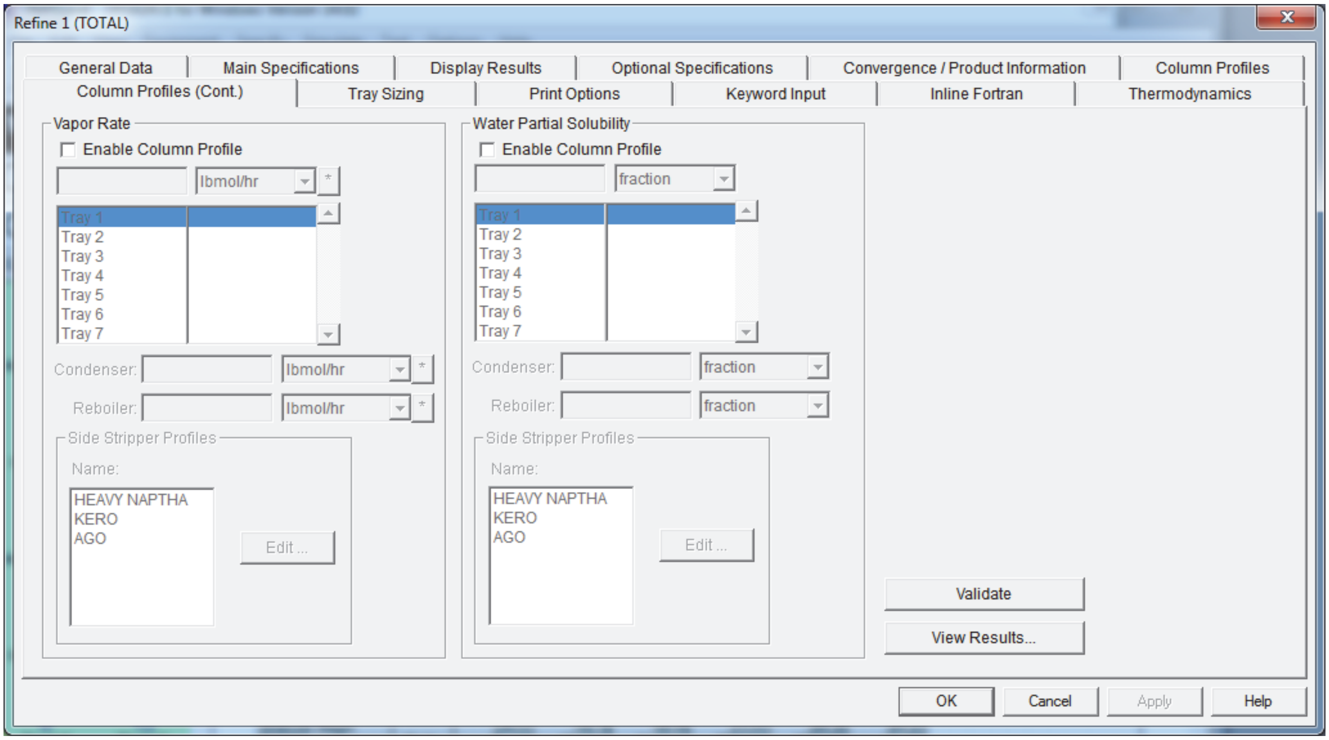


Figure 31b: Column Profiles Tab (Cont.) (from refi1.psd)

Vapor Rate

Refine automatically sets up vapor and liquid profiles based on constant molar overflow using your initial product guesses or a Fenske shortcut calculation (if no guesses were provided), and feed information. Heat removals/additions and chimney trays can introduce significant non-linearity in a vapor profile. The values that you enter for the vapor profile are initial guesses only. These values may be adjusted throughout the Refine column calculations.

Any zero values entered for the main column or side stripper will be linearly interpolated by the simulator, except for the first and last values. The interpolation will start at the second stage of the main column or side stripper and end at the next to last stage of the main column or side stripper.

Water Partial Solubility

Enter a value for the main column reboiler, if any, only if no additional water decants (LOC SIDe WATer) have been specified; condensers always decant free water.

Water partial solubility refers to the fraction of "free" water on each tray with respect to the bulk liquid on the tray (water and hydrocarbons). For quench columns, these values will typically be in the range of .97 to .999. The values that you enter in the Water Partial Solubility profile are initial estimates and may be changed during Refine column calculations. You should enter a zero for the first tray after the last water side draw product, the remaining trays will use 0.0 also.

NOTE: This profile is required for Water Pumparounds and Water Side draws.

Optional Specifications: Tray Sizing

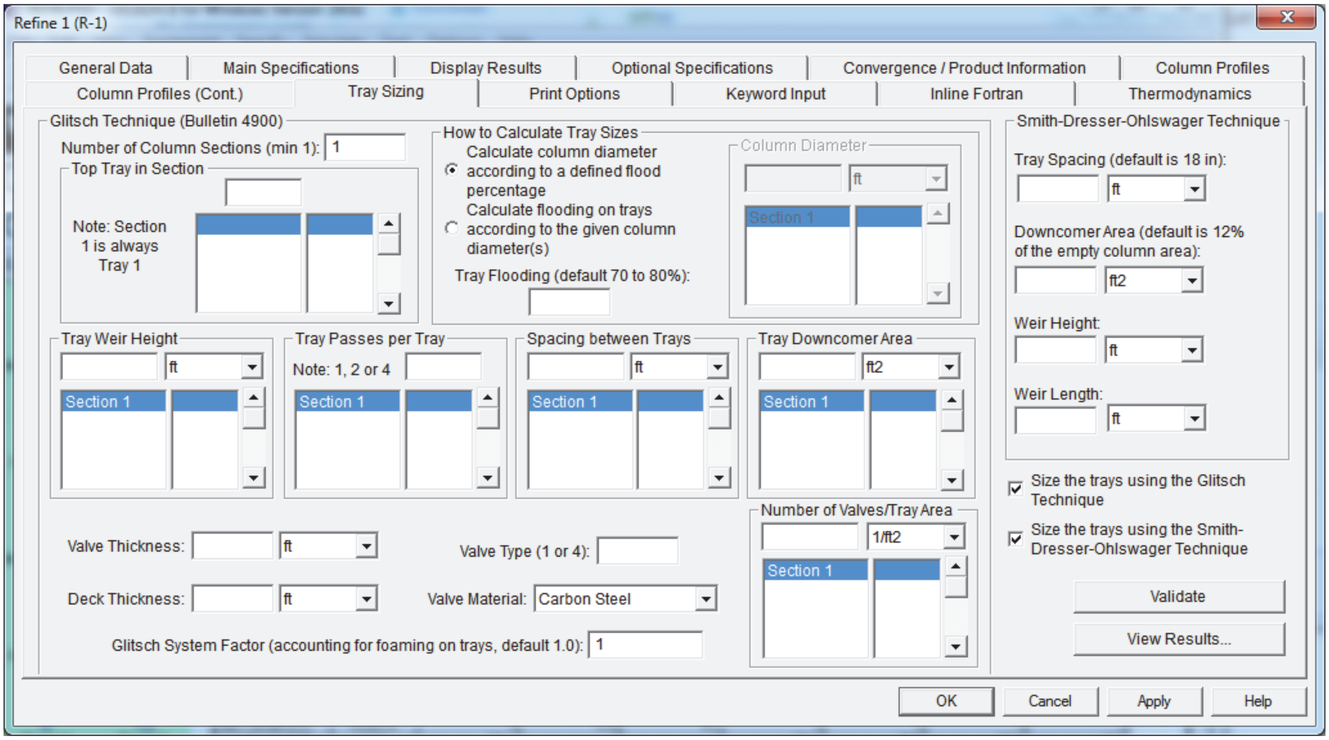


Figure 32: Tray Sizing Tab (from refi6.psd)

There are two different shortcut sizing techniques in the Refine column calculations. Both calculations are performed automatically using default values unless you enter one or more of the data items listed below.

Smith-Dresser-Ohlswager Technique

The first is the Smith-Dresser-Ohlswager technique (Smith, R.B., Dresser, T. and Ohlswager, T., Hydrocarbon Processing, Vol. 40, No. 5., pp. 183- 184 (1963)). This correlation calculates the column diameter at 100 percent of flood.

All commands are optional. If you do not want to use the default 18 inches settling height, you must enter all of the following commands:

Data Item	Description
Tray Spacing	Enter the tray spacing to be used along with Weir Height and Weir Length in calculating the settling height. Default is 18 inches settling height.
Downcomer Area	Enter the downcomer area in square feet. Default is 12 percent of the empty column area.
Weir Height	Enter the Weir Height to be used along with Tray Spacing and Weir Length in calculating the settling height.
Weir Length	Enter the Weir Length to be used along with Tray Spacing and Weir Height in calculating the settling height.

Values entered with the Tray Spacing, Weir Height, and Weir Length data items are used in calculating the settling height (tray spacing minus clear liquid depth). If any one of these three data items is not entered, the settling height is assumed to be 18 inches. Minimum settling height is 2 inches and maximum is 30 inches for the correlation. Downcomer Area is used in calculating the vapor velocity on each stage. If not entered, the Downcomer Area is set to 12 percent of the empty column area.

Glitsch Technique

The second sizing technique is provided in cooperation with Glitsch Incorporated (Glitsch Bulletin 4900, Design Procedure for Ballast Trays). This shortcut method was developed for ballast trays but is also applicable for sieve trays with 14% hole area. This technique is applicable for tray spacings of 12-48 inches and percent of flood values from 20-100%.

By default the Glitsch, Inc. correlation calculates column tray diameters at 80% of flood based on tray loadings, system factor and tray spacing. The number of passes will be calculated, if you do not specify a value. You can specify a different percent of flood using the Glitsch Percent flood command.

Data Item	Description
Number of Column Sections	Enter the number of column sections to use. The minimum number is 1.
How to Calculate Tray Sizes	Select either: Calculate column diameter according to a defined flood percentage Calculate flooding on trays according to the given column diameter(s) If you selected Calculate column diameter according to a defined flood percentage: Tray Flooding (default 70 to 80%): Enter the percent of flood for the column. If you selected Calculate flooding on trays according to the give column diameter(s): Click on the name of a section from the list and enter a column diameter then select a unit. Repeat this for each section.
Tray Flooding	Enter the percent of flood for the column. If not specified, the program will calculate. One input value allowed.
Tray Weir Height	Click on the name of a section from the list and enter a weir height then select a unit. Repeat this for each section.
Tray Passes per Tray	Click on the name of a section from the list and enter the number of tray passes. One input value is allowed: 1, 2, or 4 passes.
Spacing between Trays	Enter the spacing(s) of trays. Default is 2 feet. The technique is valid for tray spacings from 1 to 4 feet.
Tray Downcomer Area	Enter the downcomer area in square feet. Default is 12 percent of the empty column area.
Number of Valves/Tray Area	Click on the name of a section from the list and enter the number of valves per square foot of active area and select a unit. Repeat this for each section.
Valve thickness	Enter the valve thickness and select a unit.
Valve Type	Enter the valve type. Either 1 or 4 is available (per Glitsch Bulletin 4900). 1 serves as a general purpose standard size unit, used in all services. The legs are formed integrally with the valve for deck thicknesses up to 3/8". 4 signifies a venturi-shaped orifice opening in the tray floor which is designed to reduce substantially parasitic pressure drop at the entry and reversal areas. A standard ballast unit is used in this opening, normally. The maximum deck thickness permissible with this opening is 10 gage.
Deck Thickness	Enter the deck thickness and select a unit.
Valve Material	Select one of the following: Aluminum Carbon Steel Copper Monel Nickel 200 Hastalloy Stainless Steel 304 Lead Titanium
Glitsch System factor	Enter the system factor for foaming on the trays. Default is 1.0 which is appropriate for non-foaming, regular systems. One input value allowed.

Chapter 6.35

Print Options

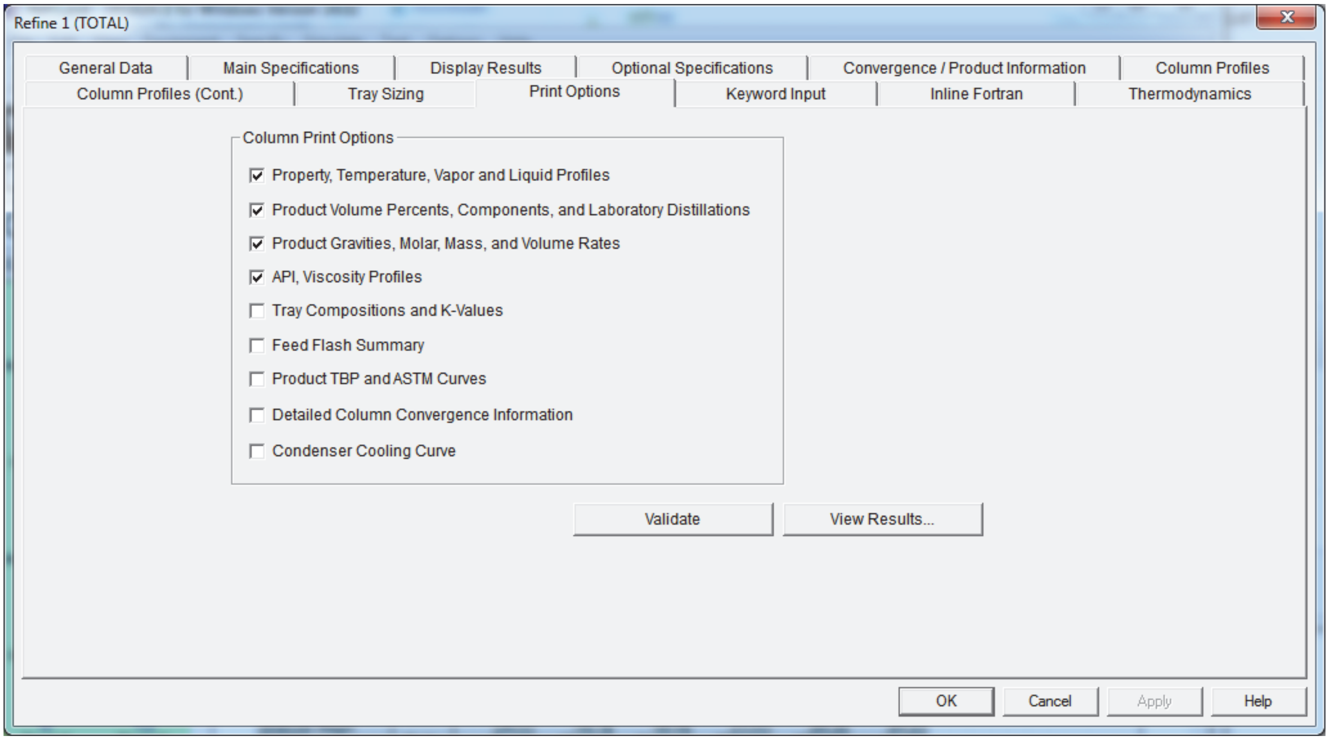


Figure 33: Print Options Tab (from refi1.psd)

Use this tab to control the print options of the Refine column. You should make a selection in each of the fields on this dialog.

The print option default is ON for:

1. The Property, Temperature, Vapor and Liquid Tray-By-Tray Profiles;
2. The Product Volume Percents, Components, and Laboratory Distillations for Products and Pumparounds;
3. The Product Gravities, Molar, Mass, and Volume Rates;
4. The API and Viscosity Tray-By-Tray Profiles;

The print option default is OFF for:
1. Tray Compositions and K-Values;
2. Feed Flash Summary;
3. Product TBP and ASTM Curve Plots;
4. Detailed Column Convergence Information;
5. Condenser Cooling Curve.

The Condenser Cooling Curve print option is available only for Refine columns with condensers.

Data Item	Description
Property, Temperature, Vapor and Liquid Profiles	Check the box to turn this print option on.
Product Volume Percents, Components, and Laboratory Distillations	Check the box to turn this print option on.
Product Gravities, Molar, Mass, and Volume Rates	Check the box to turn this print option on.
API and Viscosity Profiles	Check the box to turn this print option on.
Tray Compositions and K-Values	Check the box to turn this print option on.
Feed Flash Summary	Check the box to turn this print option on.
Product TBP and ASTM Curve Plots	Check the box to turn this print option on.
Detailed Column Convergence Information	Check the box to turn this print option on.
Condensing Cooling Curve	Check the box to turn this print option on.

Optional Specifications: Keyword Input

Click the Keyword Input tab. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II Reference Guide.

Refine Distillation Columns Examples

There are several sample flowsheets in “Chapter 40: Refine Column Samples - c:\designii\samples\equipmnt\refine” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.36: Shortcut Fractionator

General

The Shortcut Fractionator module simulates a simple distillation column with one feed and a top and bottom product. For a specified recovery of light and heavy key components, the following is estimated:

feed tray location
reflux ratio
number of theoretical stages
condenser duty
reboiler duty

This information is reported for six standard percentages of minimum reflux (105,110,120,140,170, 200%) and for another percent of minimum reflux you can specify.

Details

An essentially unlimited number of input streams can be coded; they will be adiabatically mixed to the lowest feed pressure. Two output streams (a top and bottom product) must be coded.

The column is assumed to have both a condenser generating reflux, and a reboiler generating vapor. Only a top product (liquid or vapor) and a bottoms liquid product are calculated. To simulate a column with a vapor and a liquid overhead product run the column as a PARtial condenser and as a TOTal condenser and interpolate between the two results.

For other columns not having both a condenser and reboiler, you can use the COMponent SPLitter. To simulate columns with multiple products, break the column into sections each having two products, and simulate the sections individually.

The SHOrtcut fractionator results are most reliable for "ideal" separations. The accuracy and reliability of the results will decrease as the degree of nonideality increases.

Method of Calculation
The SHOrtcut fractionator uses standard text book techniques in calculating feed tray location, reflux ratio, theoretical stages, and condenser and reboiler duties. The relative volatilities of the components are assumed to be constant throughout the column. The vapor and liquid rates above the feed tray are assumed constant, as are the vapor and liquid rates below the feed tray. The following equations are used:

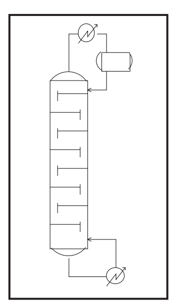
Shiras	for calculation of component distribution between the top and bottom product for components other than the light and heavy key.
Underwood	for minimum reflux ratio.
Fenske	for minimum theoretical stages.
Erbar-Maddox	for theoretical stages if the feed is between bubble point and dew point conditions.
Gilliland	for theoretical stages if the feed is subcooled or superheated.
Kirkbride	feed tray location (from top of column).

See sample printout in Example 1 that includes a plot of reflux versus stages required for separation.

Please see the online **DESIGN II** Help, topic ***Equipment/Shortcut Fractionator*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 34: Shortcut Fractionator*** for more details.

Symbols

The Shortcut Fractionator unit module has one symbol:



The Shortcut Fractionator module requires that at least one inlet stream and two outlet streams be connected to the module.

The first outlet stream will be connected to the top of the condenser if it is a partial condenser. Otherwise if it is a total condenser; then the first outlet stream will be connected to the bottom of the condenser.

The second outlet stream will always be connected to the reboiler.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

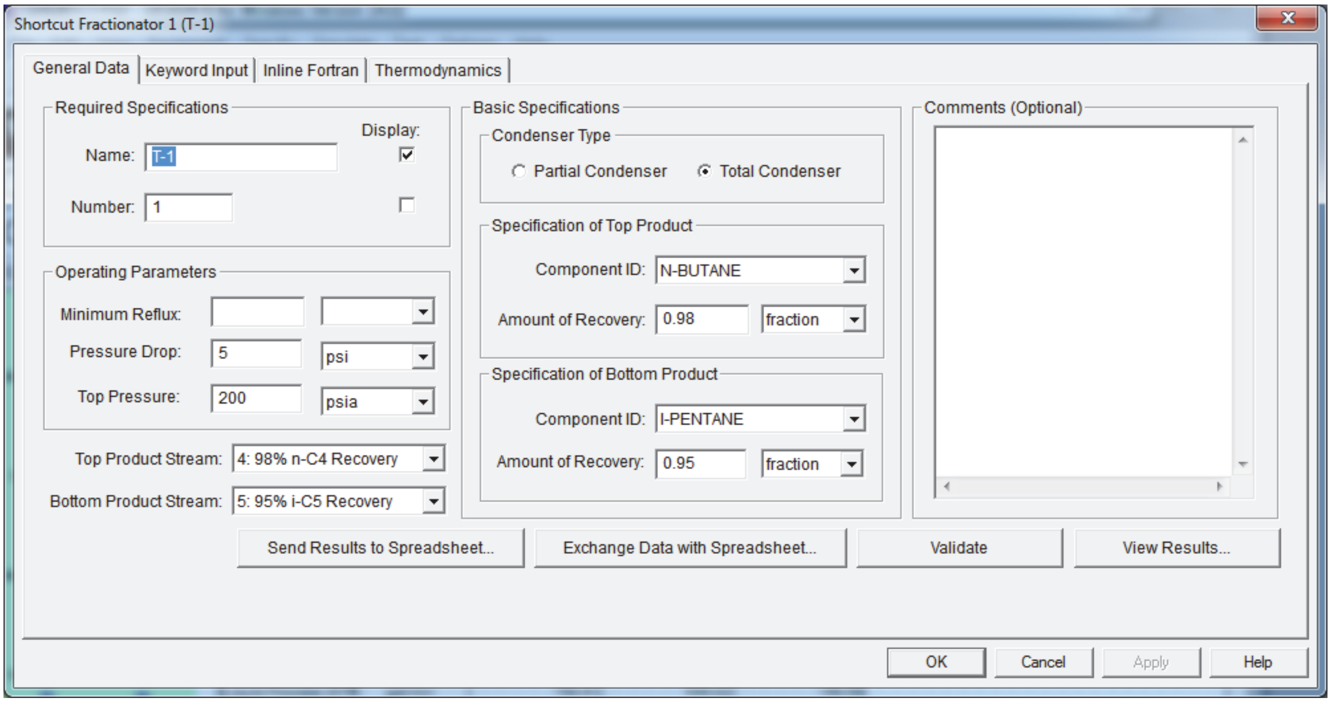


Figure 1: Shortcut Fractionator Dialog (from shortct.psd)

General Data

Data Items	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment, ; then choose to display it on the flowsheet.
Condenser Type	
Partal condenser	Specifies vapor top product.
Total condenser	Specifies liquid top product.
Specification of Top Product/Bottom Product	
Component ID	From the drop-down list, select the light key component in the feed that is to go to the top product (or bottom product).
Amount of Recovery	Enter the fractional amount of the selected heavy key component.

Neither 100% nor 0% recoveries can be specified (.99999 and .00001 are the mole fraction limitations). The light and heavy key components do not have to be next to each other in the order of components, nor does the light key have to be first in order. The light and heavy key component separation specified must be realistic. At least 50 percent of the light/heavy key component must go to overhead/bottoms.

Operating Parameters	
Minimum Reflux	Enter the percent of minimum reflux. Default is 130.
Pressure Drop	Enter the column pressure drop. Default is zero.
Top Pressure	Enter the pressure for the top product. Default is: (feed pressure - 0.5 x DEL.)
Top Product Stream	Open the drop down list and select a stream to designate as the Top Product Stream.
Bottom Product Stream	Open the drop down list and select a stream to designate as the Bottom Product Stream.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. ny line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II Reference Guide.

Shortcut Fractionator Examples

There are several sample flowsheets in “Chapter 41: Shortcut Fractionator Column Samples - c:\designii\samples\equipmnt\shortcut” of the DESIGN II for Windows Tutorial and Samples Guide.

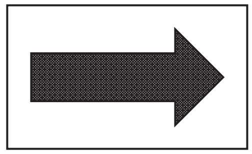
Chapter 6.37: Sink 1 and Source 1

General

The Sink 1 and Source 1 symbols are visual representations only; these symbols currently have no functionality.

Symbols

Sink 1 and Source 1 use the same symbol:



Chapter 6.38: Spreadsheet

General

There is a new dialog for the new Spreadsheet unit module.

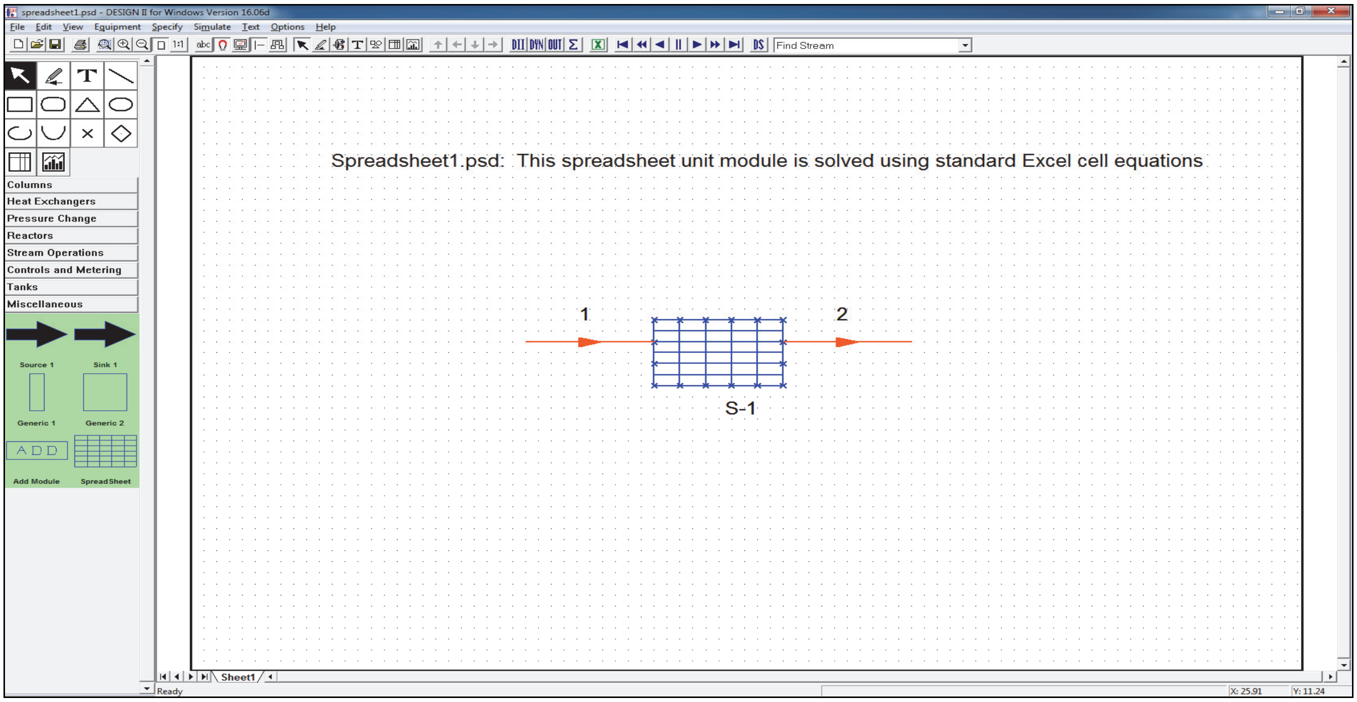


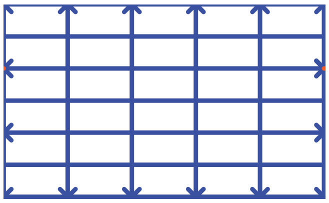
Fig. 1 Spreadsheet Unit Module

Details

The general tab of the Spreadsheet unit module dialog allows specification of the unit module name and number. The number is required to be unique for the flowsheet. The spreadsheet dimensional units for data transfer are selectable for the temperature, pressure, enthalpy, quantity, and time. The spreadsheet addresses tab allows the specification of the Microsoft Excel spreadsheet file name, the cell addresses to transfer the inlet stream data to, the cell addresses to transfer the outlet stream dat from, the type of outlet stream data to get from the spreadsheet, the type of spreadsheet calculations (cell macro or Visual Basic macro), and whether or not to dismiss the spreadhseet when finished..

Symbols

This unit module has one symbol.



The symbol looks like a table object with many stream connection points.

Properties

When you double click on the symbol, the general data dialog for this symbol will pop up.

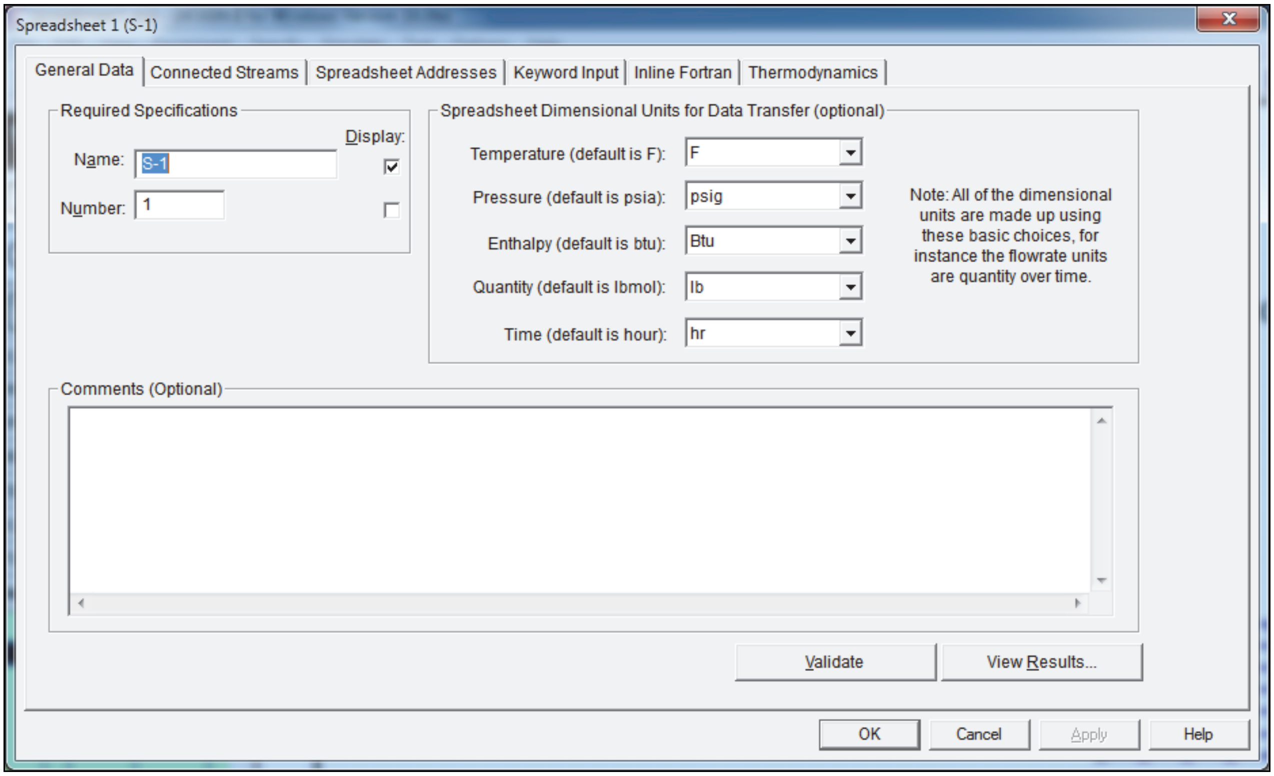


Fig. 2 General Data Dialog

The general tab of the dialog allows specification of the unit module name and number. The number is required to be unique for the flowsheet. The spreadsheet dimensional units for data transfer are selectable for the temperature, pressure, enthalpy, quantity, and time.

General Data

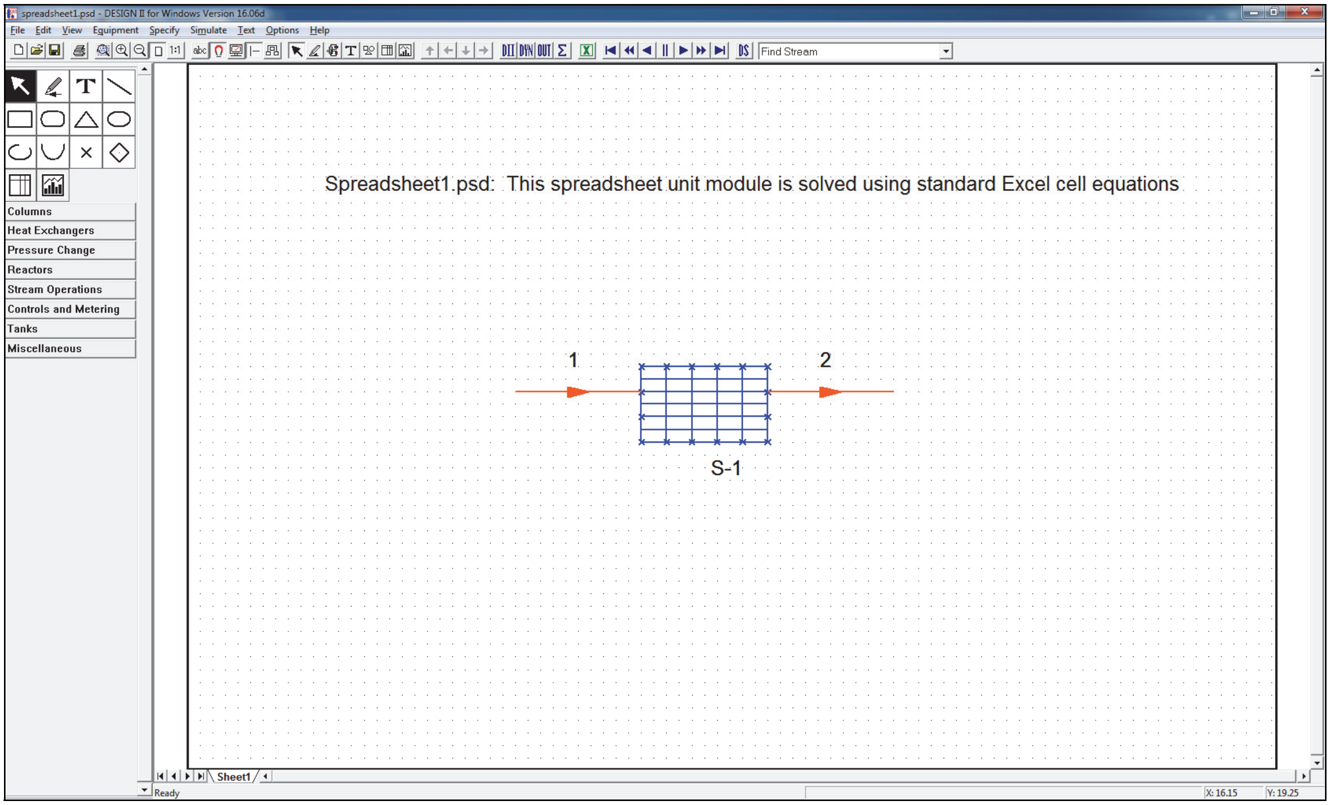


Fig. 3

Connected Streams

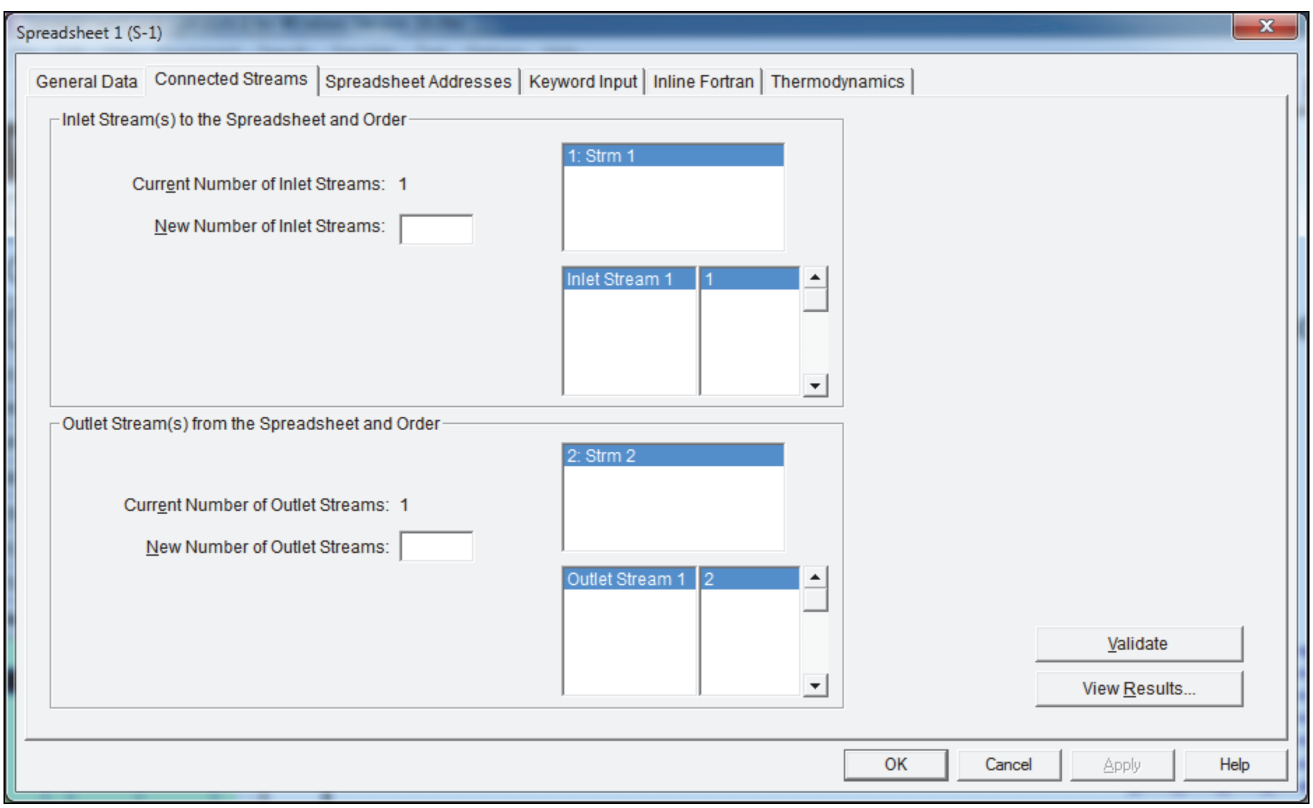


Fig. 4 Connected Streams Dialog

Spreadsheet Addresses

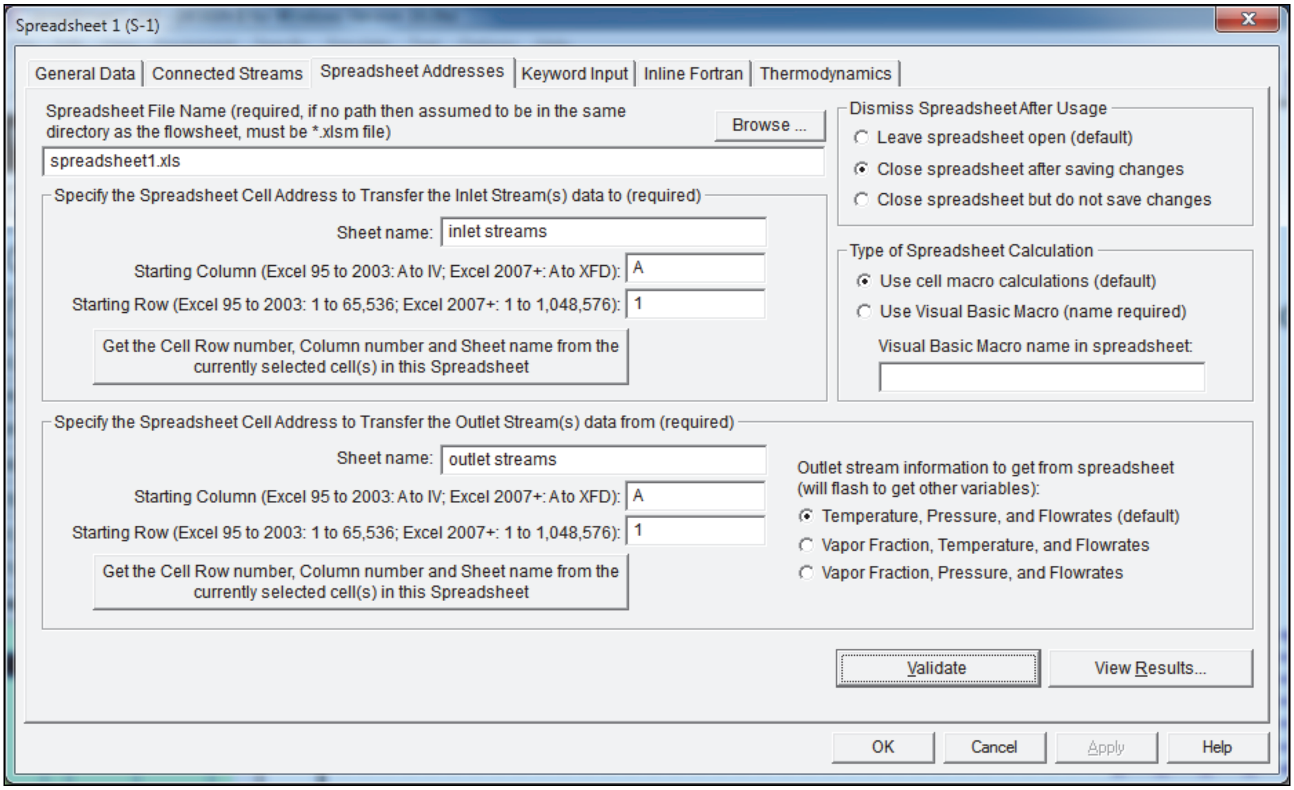


Fig. 5 Spreadsheet Addresses Dialog

The addresses tab allows the specification of the Microsoft Excel spreadsheet file name, the cell addresses to transfer the inlet stream data to, the cell addresses to transfer the outlet stream dat from, the type of outlet stream data to get from the spreadsheet, the type of spreadsheet calculations (cell macro or Visual Basic macro), and whether or not to dismiss the spreadhseet when finished.

Required Specifications: Keyword Input

Click the Keyword Input tab. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Spreadsheet Examples

There are several sample flowsheets in “Chapter : Spreadsheet Samples - c:\designii\samples\equipmnt\spreadsheet” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.39: Stream Manipulator

General

Stream Manipulator is a module that duplicates/manipulates one stream to two streams or calculates heating value.

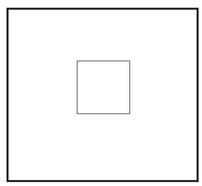
Details

The Stream Manipulator module can be used to calculate a stream property (such as heating value) on a molar or standard vapor basis (e.g. BTU/SCF). The module can also be used to modify stream component flowrates by a single factor for all components or by an individual factor for each component. The component flowrates can be added, subtracted, multiplied, or divided by the factors. The first stream is a copy of the input. When two outlet streams are specified, the second outlet stream is the modified stream.

Please see the online **DESIGN II** Help, topic ***Equipment/Stream Manipulator*** or the **DESIGN II** Unit Module Reference Guide ***Chapter 35: Stream Manipulator*** for more details.

Symbols

This module has one symbol:



The Stream Manipulator module requires that at least one inlet stream and at least one outlet stream be connected to the module.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

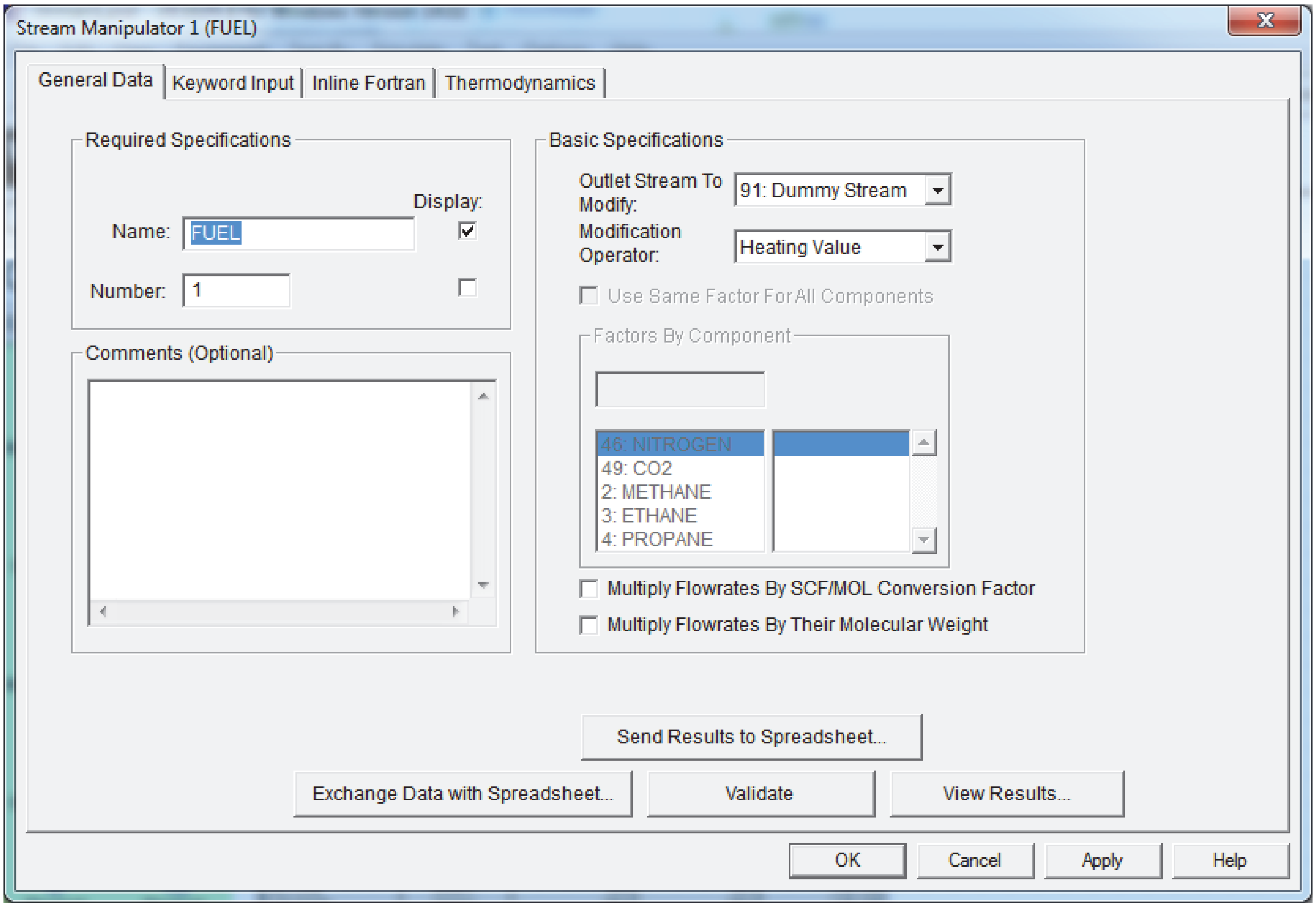


Figure 1: Stream Manipulator Dialog (from strman1.psd)

General Data

Data Items	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment; then choose to display it on the flowsheet.
Outlet Stream to Modify	Select an outlet stream from the drop-down list.
Modification Operator	Select one of the following modification operators: Add: Add the factor to component flowrates. Subtract: Subtract the factor from component flowrates. Multiply: Multiply the factor with component flowrates. Divide: Divide the factor by component flowrates. Heating Value: Select this command to get the gross heating value calculated for a hydrocarbon product stream. No other calculation option is allowed. **NOTE**: DESIGN II has gross heating values for most components and net heating value for 65 components (Fig. 23-2 of page 23-4 in GPSA manual). Net heating values for components not covered by data are estimated from gross heating values by subtracting water latent heats for estimated amount of water produced. DESIGN II also estimates heating values for petroleum fractions/cuts. You may enter HIGh and LOW HEAting values for petroleum fractions, or an estimate for CARbon to HYDrogen weight ratio for calculating the gross heating value.
Use Same Factor for All Components	Select this checkbox to enter one factor to use for all the components.
Factors By Component	If you do not select Use Same Factor for All Components, click on the name of the component from the list and type a factor for that component. The lower limit is 1E-10 or any greater value (such as 1.01E-10). There is no upper limit.

Multiply Flowrates by SCF/MOL Conversion Factor	If you selected Multiply as the Modification Operator, you can select this checkbox to multiply all component flowrates by a SCF/MOL conversion factor, in addition to being modified by the factor you entered.
Multiply Flowrates by Their Molecular Weight	If you selected Multiply as the Modification Operator, you can select this checkbox to multiply all component flowrates by their molecular weights, in addition to being modified by the factor you entered.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II Reference Guide.

Stream Manipulator Examples

There are several sample flowsheets in “Chapter 42: Stream Manipulator Samples - c:\designii\samples\equipmnt\strman” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.40: Tank

General

The Tank module calculates the dynamic behavior of feed, accumulations, and products from a constant volume tank. Temperature, pressure, and compositions will be calculated over a period of given time. Both pure components and mixed phases can be handled by the tank.

Details

For a tank, you can execute a dynamic simulation using a Time Step and Time Duration (Specify/Dynamic Settings). The initial time step calculates the initial amount of material in the tank.

Subsequent time steps and additions from feed streams (if any), subtractions from product streams (if any), will calculate the resulting temperature, pressure and compositions of the tank. If there is a vent specified then the vent stream will allow vapor to bleed off to meet the vent pressure while maintaining adiabatic equilibrium.

The product streams can be started and/or stopped according to the user's time specifications. The liquid product streams will only run, if there is liquid in the tank and the vapor product streams will only run if there is vapor in the tank.

For example:

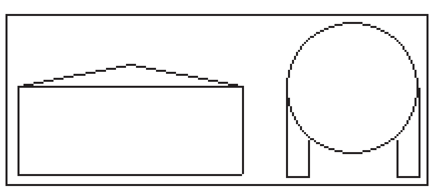
```
Product #1, start at 1 hr, stop at 3 hr
Product #2, start at 0 hr, stop at 4 hr
pr#1        <-------------->
pr#2 <---------------------------->
HR   +------+------+------+------+
     0      1      2      3      4
```

The product will only run if the start time is less than the current time and the stop time is zero or greater than the last time step value.

Please see the online ***DESIGN II*** Help, topic ***Equipment/Tank*** for more details.

Symbols

The Tank unit module has two symbols: Tank-Vertical and Tank-Spherical.



The Tank module requires that one inlet and one outlet stream be connected to the module.

General Data

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

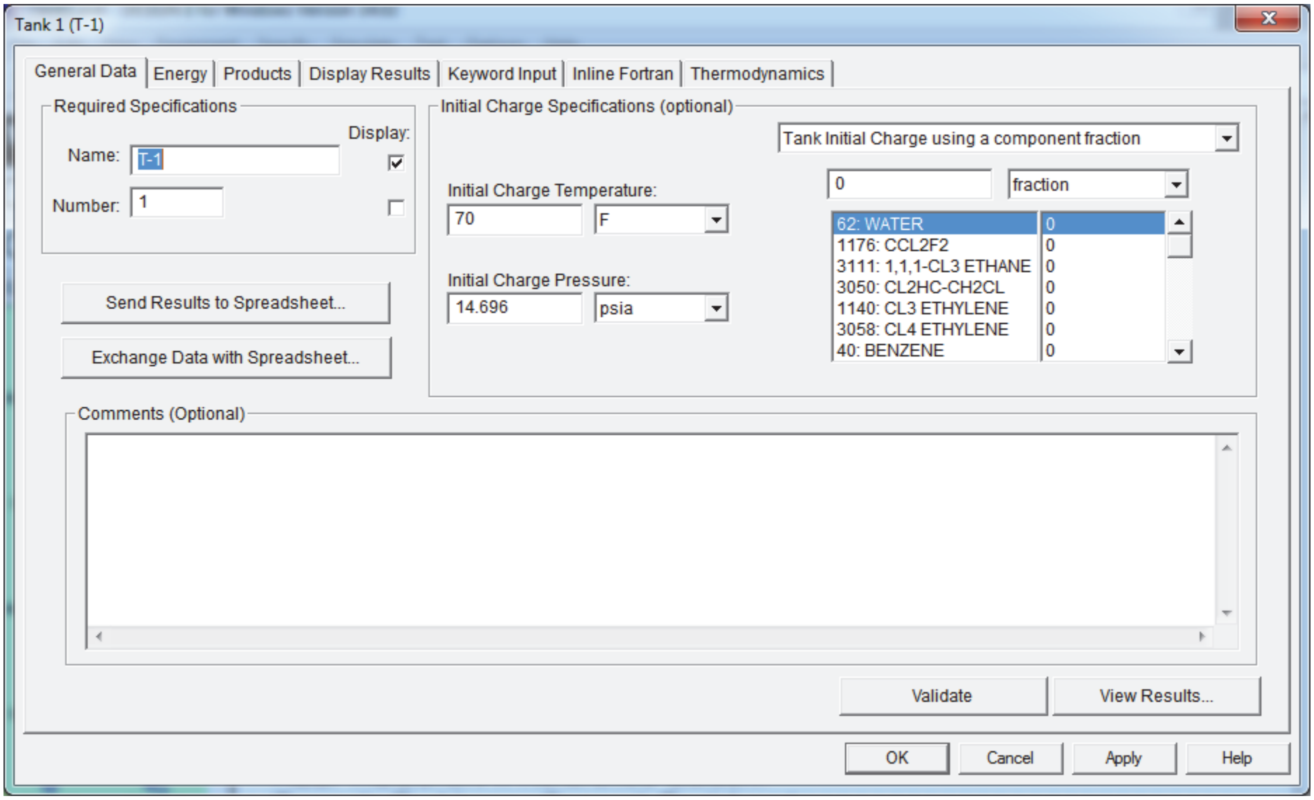


Figure 1: General Data (from tank1.psd)

Data Items	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment; then choose to display it on the flowsheet.
Tank Initial Charge using a component fraction	If you select this option, the units change for the tank initial charge field. Enter the initial charge amount and select a unit.
Tank Initial Charge using a component mole or mass	If you select this option, the units change for the tank initial charge field. Enter the initial charge amount and select a unit.
Initial Charge Temperature	Enter the temperature and select a unit.
Initial Charge Pressure	Enter the pressure and select a unit.

Energy

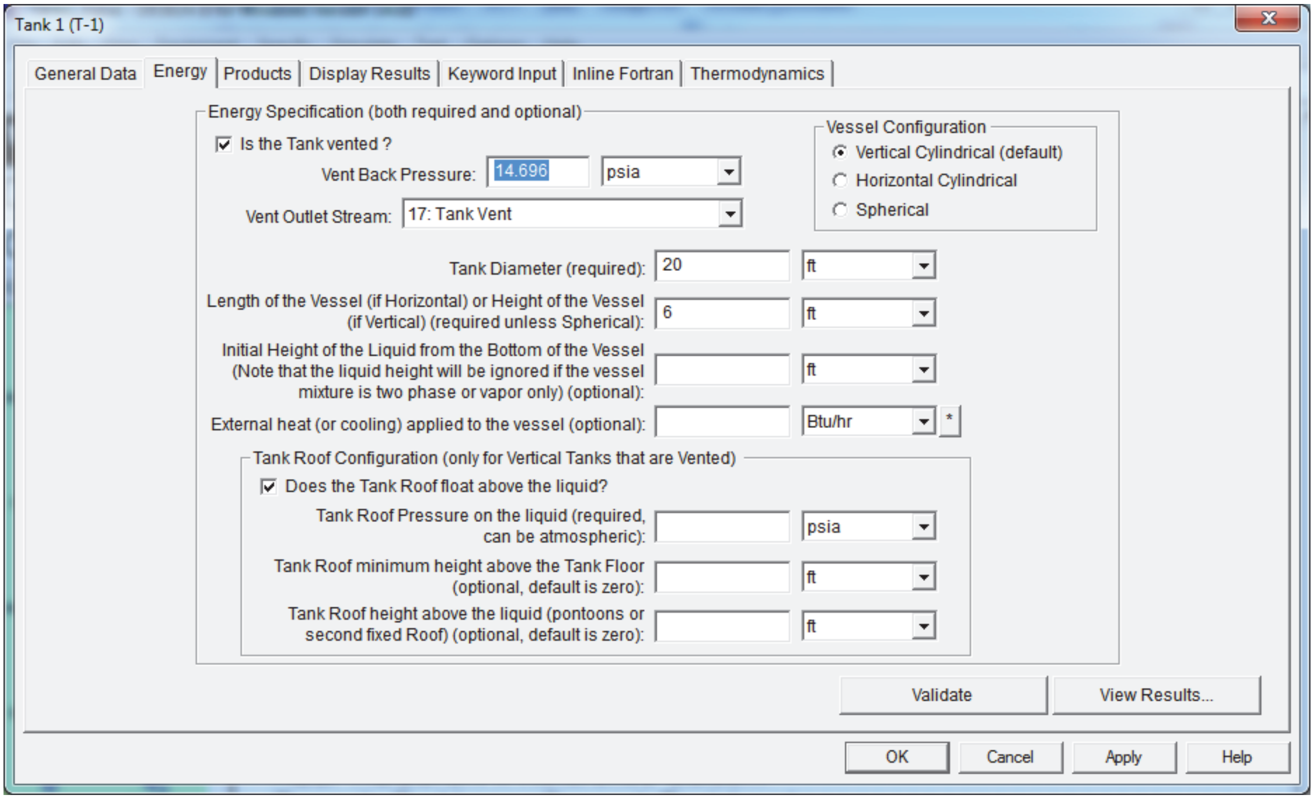


Figure 2: Energy (from tank1.psd)

Is the Tank vented	If the tank is vented, select this checkbox then enter the vent back pressure and select a unit. Next, open the vent outlet stream drop down list and select the desired stream to use.
Tank Diameter	Enter the diameter of the tank and select a unit; this is a required field.
Length of the Vessel (if Horizontal) or Height of the Vessel (if Vertical) (required unless Spherical)	Enter the measurement and select a unit; this is a required field for Horizontal and Vertical tanks.
Initial Height of the Liquid the Bottom of the Vessel	Enter the initial height of the liquid from the bottom of the tank and select a unit. This is from optional. Note: This value is ignored if the vessel mixture is two phase or vapor only. The tank product is specified on the Product tab.
External heat (or cooling) applied to the vessel	Enter the temperature applied to the tank and select a unit. This is optional.
Vessel Configuration	Select the type of tank, either vertical (default), horizontal, or spherical.

Products

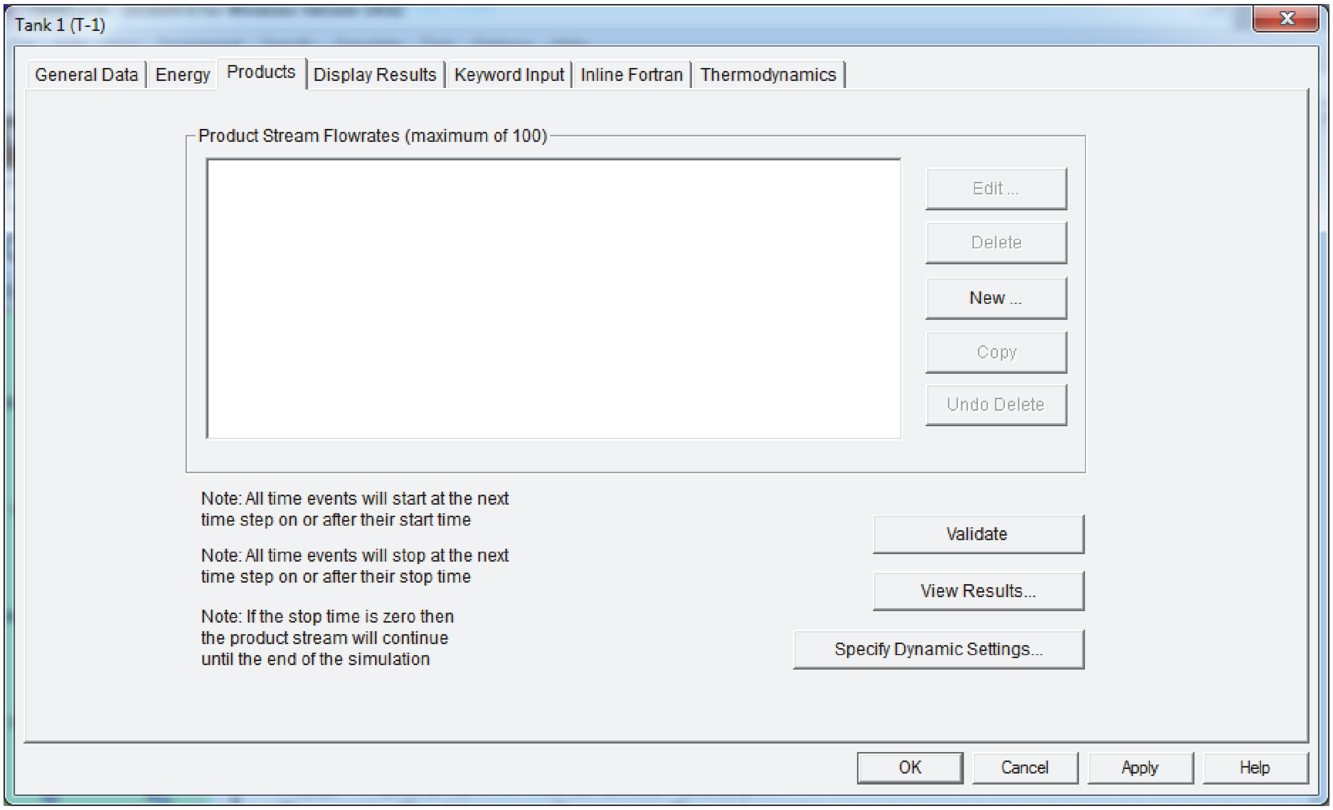


Figure 3: Products (from tank1.psd)

Use this tab to enter the product specifications for a tank.

Product Stream # 1

Flow Rate	Enter the flow rate for the first product stream and select a unit.
Stream Number	Open the drop down list and select the desired stream.
Type	Select the product for the tank from the drop down list, either Liquid or Vapor.
Start Time	Enter a start time for the product and select a unit; all time events will start at the next time step on or after their start time.
Is the Tank Vented?	Enter a stop time for the product and select a unit; all time events will stop at the next time step on or after their start time. If the stop time is zero; then the product stream will continue until the end of the simulation.
Product Stream #2-#5	Repeat this process for up to four more product streams.
Specify Dynamic Settings;	Click this button to display the General Data - Dynamic Settings to enter time step and time duration values for running a dynamic simulation. This dialog is covered in Chapter 10 Simulation.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Tank Examples

There are several sample flowsheets in c:\designii\samples\equipmnt\tank" of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 6.41: Valve

General

The Valve module performs an adiabatic (constant enthalpy) pressure reduction of a stream.

Details

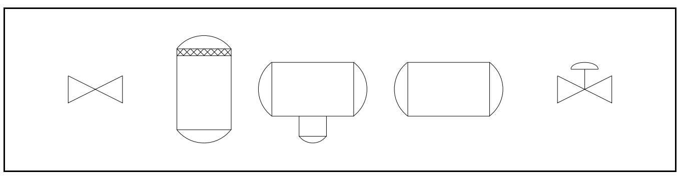
The Valve module is used for adiabatic (constant enthalpy) pressure reduction of a stream that can be vapor, liquid, or two-phase. It reports the resulting outlet temperature. Only one inlet stream and one outlet stream can be connected to this module. If phase separations are desired for the outlet stream, you can use the Flash module.

Dew and Bubble point pressure specifications can be made for the outlet stream. The module will determine the dew or bubble point pressure at a temperature you specify. This pressure will be used for the outlet pressure specification. The resulting outlet temperature is then calculated.

Please see the online ***DESIGN II*** Help, topic ***Equipment/Valve*** or the ***DESIGN II*** Unit Module Reference Guide ***Chapter 36: Valve*** for more details.

Symbols

The Valve unit module has five symbols: Valve, Valve 1, Valve 2, Valve 3 and Valve 4.



The Valve module requires that one inlet and one outlet stream be connected to the module. You can use the Flash module for phase separations of mixtures in the outlet stream.

Properties

When you double click on the symbol, the equipment properties dialog for this symbol will pop up.

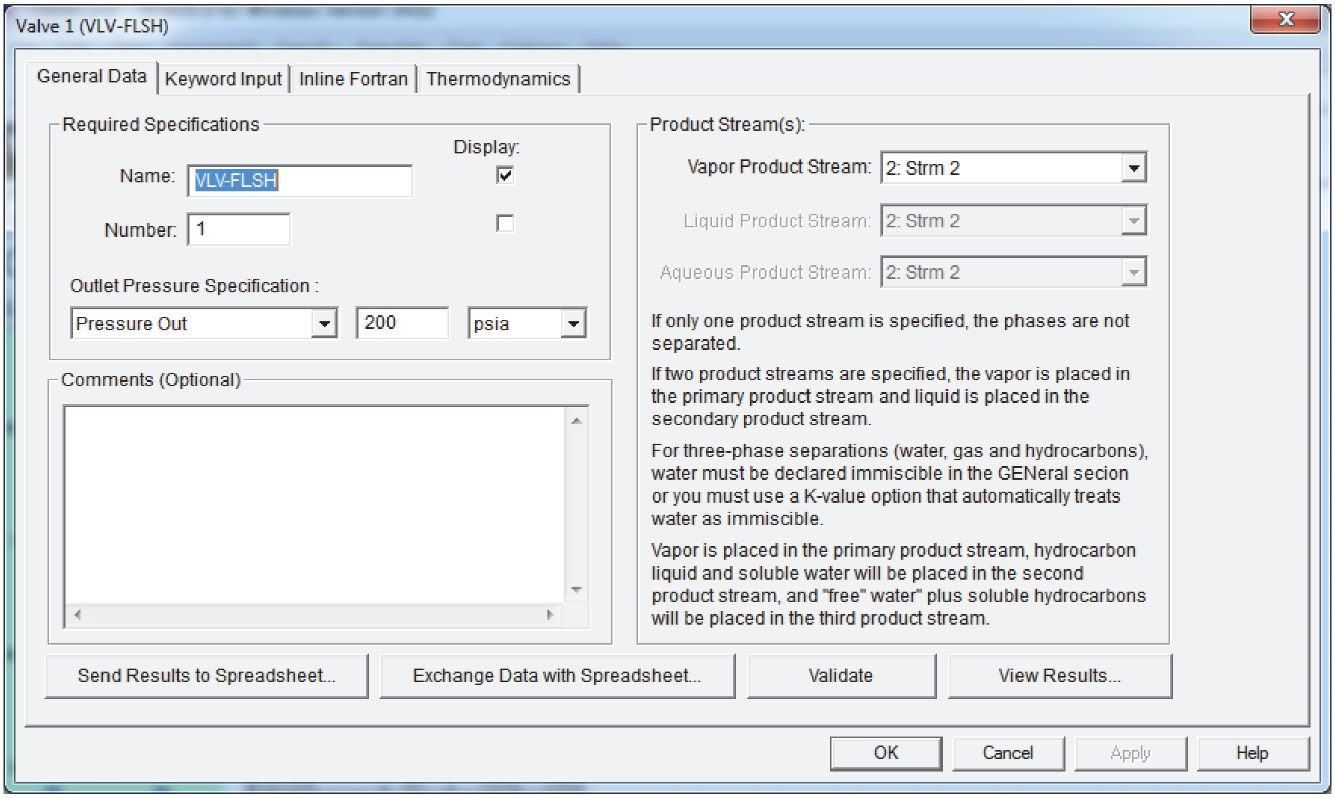


Figure 1: Valve Dialog (from valve1.psd)

General Data

Data Items	Description
Name/Number/Display	The name/number associated with the equipment. You can enter a name/number for the equipment ; then choose to display it on the flowsheet.
Outlet Pressure Specification	Pressure Drop: The pressure drop between the outlet and inlet streams. Pressure Out: The pressure of the outlet stream. Bubble Point Temperature: Specifies that the bubble point pressure be calculated at the temperature you specify. This pressure is then used as the outlet stream pressure specification. Dew Point Temperature: Specifies that the dew point pressure be calculated at the temperature you specify. This pressure is then used as the outlet stream pressure specification. Liquid CV Coefficient: An experimentally determined factor that indicates the flow capacity of a liquid (incompressible flow) during non-choked, non-flashing flow through a valve with unit differential pressure. Only U.S. System units are allowed (gpm/psi).
Vapor/Liquid/Aquous Product Stream	Open the drop down list and select the desired stream to designate as the Vapor, Liquid, or Aqueous Product Stream. If you only specify one product stream, the phases are not separated. If you specify two product streams, the vapor is placed in the primary product stream and the liquid is placed in the secondary product stream. For three-phase separations (water, gas, and hydrocarbons), you must declare water immiscible or you must use a K-value option that automatically treats water as immiscible (see the Thermodynamics tab). Vapor is placed in the primary product stream, hydrocarbon liquid and soluble water will be placed in the second product stream, and "free water" plus soluble hydrocarbons will be placed in the third product stream.

Optional Specifications: Keyword Input

Click the **Keyword Input** tab. Click Load Template and follow the instructions to specify this equipment with the Keyword Input. To specify a command delete the “C-*” before it and enter the units in parenthesis and any values after the =. Any line beginning with “*” will not be written to the DESIGN II input file. However any line with a “C-” (and no *) will be written to the input file and interpreted by DESIGN II as comments.

Optional Specifications: Inline Fortran

You may enter Inline Fortran statements in the edit screen that appears in this tab. You can specify that these statements be executed either before (PRE operation) or after (POST operation) simulation of the equipment model. For instructions about the use of Inline Fortran please refer to Inline Fortran section of the on-line DESIGN II Reference Guide.

Optional Specifications: Thermodynamics

This tab provides a list of combo boxes that allow you to pick correlations for thermophysical properties that you want DESIGN II to use for your specific unit module.

Each combo box will display the default selection for a particular correlation. To change the default, select the appropriate combo box ; then select the thermophysical property correlation. For general recommendations as to the applicability of particular correlations, see the ***Chapter 7: Thermodynamics*** in the DESIGN II General Reference Guide.

Valve Examples

There are several sample flowsheets in “Chapter 43: Valve Samples - c:\designii\samples\equipmnt\valve” of the DESIGN II for Windows Tutorial and Samples Guide.

Chapter 7: Streams

Working With Streams

Streams are represented by lines that you draw on the flowsheet. These lines connect with equipment to depict material flow. They are also used to designate inlet and outlet streams for the chemical process you are modeling.

To add streams to your flowsheet, you construct lines that proceed from equipment to equipment, from inlet to equipment, and from equipment to outlet. The lines will automatically start and end on grid points. They will also automatically connect to snap points on equipment symbols when you bring the ends of the lines near the snap point. A stream can consist of one straight line segment or many connected line segments with bends in different directions

When you place a stream on the flowsheet, the program automatically assigns a name and a number to it. The streams are numbered in the sequence you place them. However, you can enter a new name or number using the Stream dialog. This dialog is displayed whenever you choose to set specifications for a stream (see the ***Setting Stream Specifications*** section later in this chapter for details).

You can set the direction of flow for material through streams using the flow arrows that are automatically added to streams. You can also set stream specifications (for temperature, pressure, etc.).

After placing a feed or recycle stream on your flowsheet, you enter specifications for it. These specifications become linked to the stream. If you copy or move the stream, the specifications will move with it.

You can also edit a stream, by either copying or deleting it (see the ***Editing Streams*** section later in this chapter for details); or by changing its specifications (see the ***Setting Stream Specifications*** section later in this chapter for details).

You can also make limited changes to two or more streams at a time. Select the streams on the flowsheet (either hold down the Ctrl key then click on each stream) ; then right click to view a menu; choose Common Stream Properties from the menu.

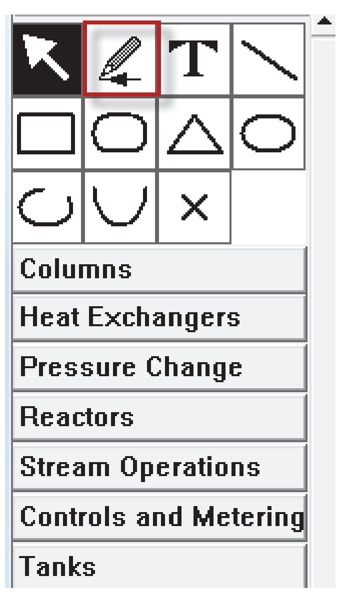
The Common Stream Properties appears, which you can use to set the display results for the selected streams (Display Results on Flowsheet provides a variety of options including Stream Name, Number, Temperature, Pressure, etc.). You can also choose the Thermodynamics tab to select common thermodynamic options for the selected streams.

Details

Please see the online DESIGN II Help topic "Stream Feeds" or the DESIGN II General Reference Guide, "Chapter 3:Specifying Feed Streams" for more details.

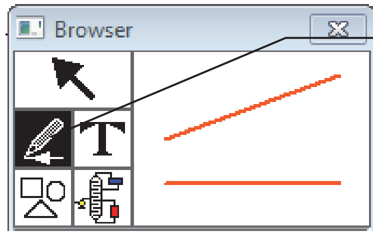
Adding Streams

To add streams to your flowsheet, you must first click on the **Stream** tool of the Toolbox (or optionally the Stream tool of the **Browser** if you are using it or the **Stream** button on the toolbar).



Stream tool on the Toolbox

Stream tool on the tool bar



Stream tool on the Browser

This places DESIGN II for Windows in its Stream mode; all functions you perform in this mode will relate to working with streams until you switch to another mode (such as Equipment or Text mode). The cursor changes to a pencil (leaning to the left) when positioned over the drawing area.

Before adding streams, you can turn on:

- Orthogonal lines (open the **Options** menu and select **Ortho Lines**).
- Line clash detection.
- Snap Points for alignment and stream placement (open the **Options** menu and select **Show Snap Points).**

To add a stream:

1. Move the pencil to the location on the drawing where you want the stream to begin and click the left mouse button. The cursor will switch to a right-leaning pencil, and you can start to draw the stream.
2. Move the mouse in the direction you want to draw the stream. A line representing the stream follows the movements of the mouse.
3. Press the left mouse button to end the current line segment for the stream.
4. To add another line segment, move the mouse in the direction you want the stream to go; a second line segment appears. By moving the mouse, you can change the direction of the line segment. This allows you to put bends in streams.
5. Option: to modify a line segment's length or direction, click the right mouse button at either end of a line. The previously drawn segment can now be repositioned by moving the mouse to a new end point. For multiple line segments, continue pressing the right mouse button to delete each segment, or to change its length and direction.

   NOTE: if you have completed the stream (the pencil cursor leans to the left), placing the pencil cursor over the stream and pressing the right mouse button will delete the stream and its associated information. You will be asked to verify the deletion.
6. To end a stream, double-click. The stream now will appear as a set of connected line segments, and has a number and name attached to it (unless you have elected to turn this display off; see ***Setting Stream Specifications***).

You can now either:

- edit the stream you just placed,
- change the flow direction using arrows,
- enter the specifications for the stream,
- add another stream (repeating the steps listed above),
- change to another mode (such as Equipment, Text, etc.)

Setting Flow Direction

Flow direction is automatically set when you draw a stream (where you start and end the stream indicates the initial flow direction; you can change the direction if necessary. The program uses the direction of flow arrows to establish that the correct number of inlet and outlet streams have been connected to an equipment symbol.

To modify the flow direction of a stream:

1. Click on the **Selection** tool of the **Toolbox**.
2. Move the cursor over the stream for which you want to change the flow direction.
3. Right click on the stream and select Reverse Stream Flow Direction from the pop up menu that appears.

Extending Streams

You can extend or lengthen an existing line as follows.

1. Locate your pencil (left leaning) cursor at the end of the existing line and click once on the left mouse key. The line will turn green and it is selected.
2. Click once on the RIGHT mouse key and the line will turn black, ready to be stretched by moving the mouse. After you reach a desired location, left click once to change direction or double click to fix the stream (line red).

Editing Streams

You can copy or delete streams from your flowsheet.

NOTE: Before editing a stream, make sure DESIGN II for Windows is in the **Stream** mode.

Moving a Stream

There are two ways to move streams, either by:

- Using the **Selection tool** to enclose the stream completely in a dashed-line box; then move the pencil cursor inside of the box and drag to a new position, or
- Moving the equipment symbol the stream is attached to, with the **Auto-Routing** option turned on.

Copying Streams

You can copy a stream (or streams) using the **Copy** function under the **Edit** Menu.

To copy a stream or streams:

1. Select the stream by clicking on it.
2. Open the **Edit** menu and select **Copy**. **NOTE**: Copied streams keep all specifications you set for them.
3. Open the **Edit** menu and select **Paste**. A copy of the stream is placed in the middle of the screen, while the original stream remains in place.

Disabling/Enabling Streams

You can disable (and enable) any stream on the flowsheet. You can also disable equipment and sheets. All validation rules apply as if the disabled stream does not exist. For instance, if a piece of equipment is disabled then all product streams from that equipment must now be specified as if they are feed streams to the process. Disabled streamd will have "Disabled" written across the stream and an X placed across it.

1. To disable/enable a stream, right-click on the stream.
2. Select Disable (or Enable) from the pop-up menu.

Deleting Streams

To delete a stream:

1. Place the pencil (left leaning) anywhere on the stream and click the left mouse button. The stream is selected.
2. Click the right mouse button. A message displays, asking if you want to delete the stream. Click **Yes** to delete it, **No** to cancel the deletion.

   NOTE: Once you delete a stream, it is permanently removed and you cannot recover it.

Setting Stream Specifications

You can enter stream data or specifications using the Stream dialog.

To set stream specifications:

1. Double-click on the stream. Or, select the stream ; then choose the **Specify Item** under the **Edit** menu. The Stream dialog displays.

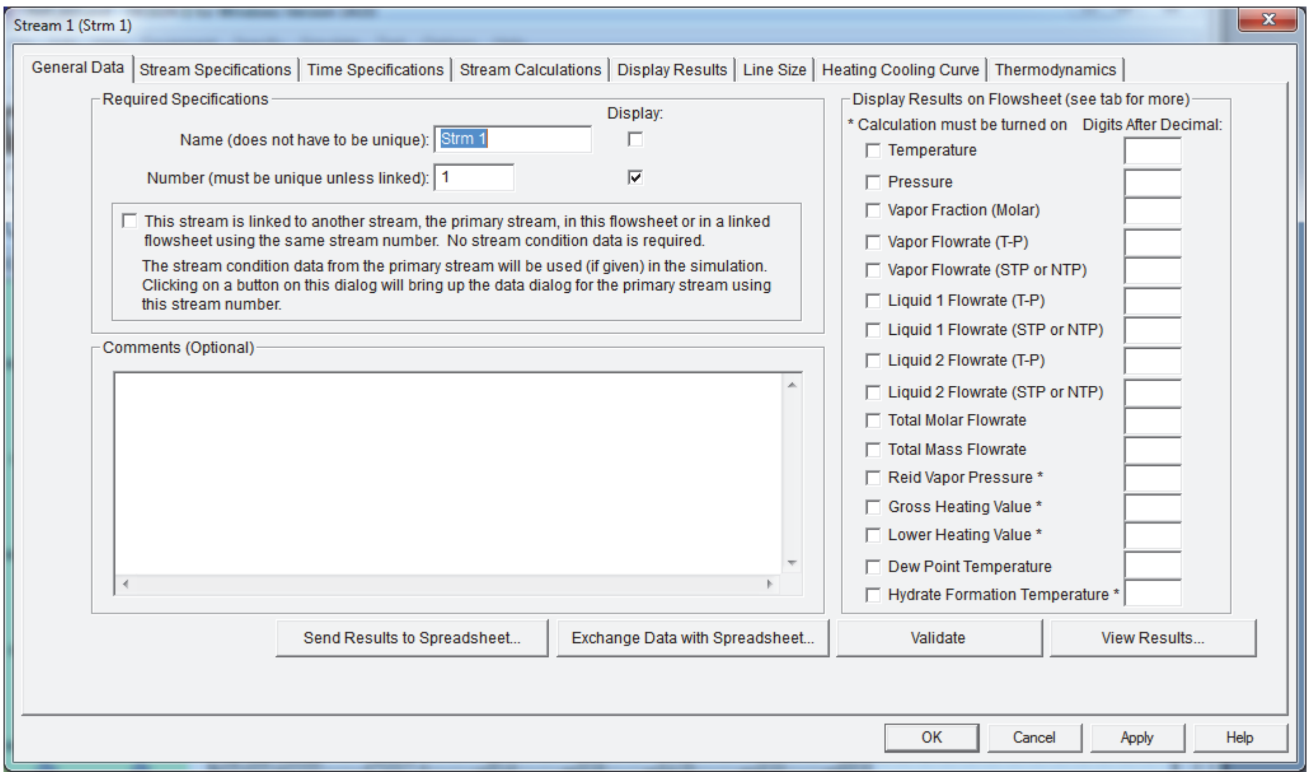


Figure 1: Example General Data tab from Stream Dialog (from refi_ex3.psd)

2. Enter a new stream name and/or number if desired**.** **Stream numbers must be unique.** You can also click the **Display** box to show/hide the name and/or number next to the stream on the flowsheet. You can select how the stream is linked, and what results to display on the flowsheet.
3. To set specifications for a stream, click on the **Stream Specifications** tab.

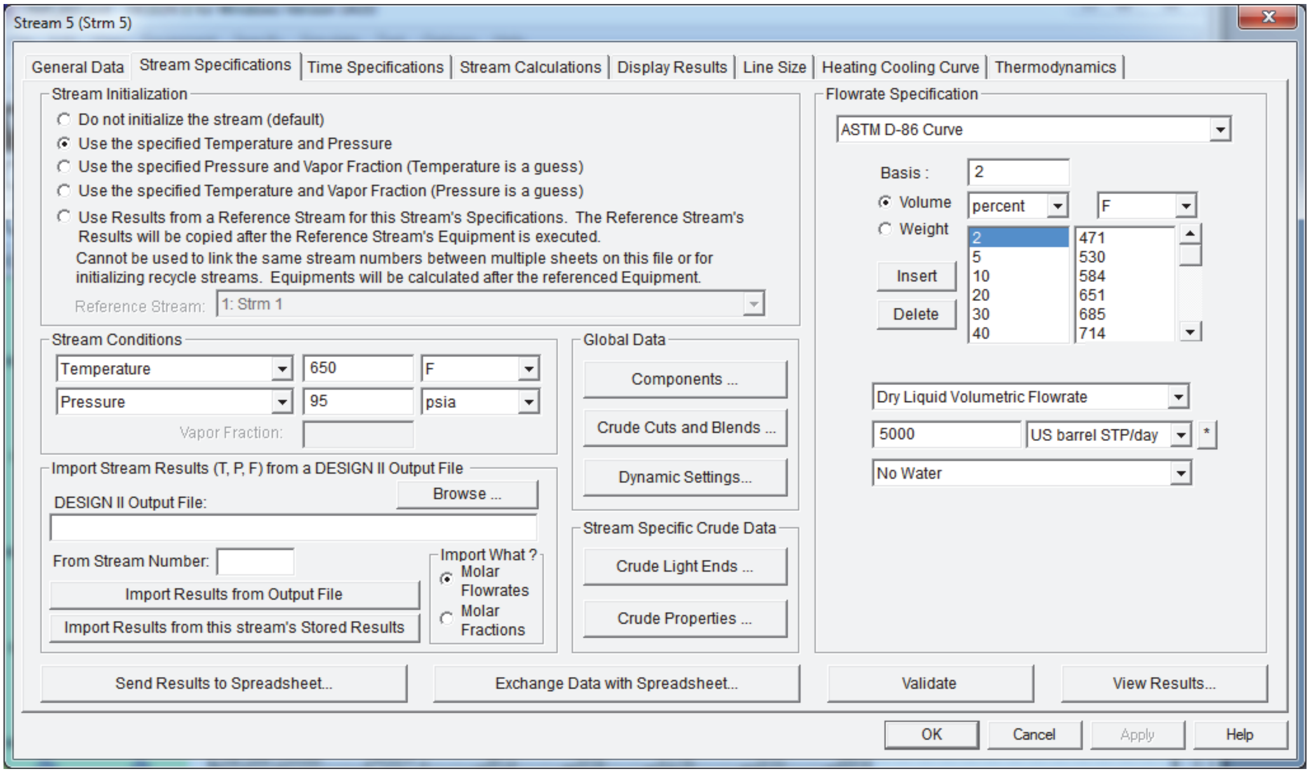


Figure 2: Stream Specifications tab from Stream Dialog (from refinery.psd)

4. Enter data for each category of entries (Temperature, Pressure and one type of Flowrate specification) on this dialog. If you need help completing this dialog, view the Help topic for the setting stream specifications by clicking the **Help** button.

   There are six group boxes for entering stream specifications or data. They are:

Stream Initialization	Specify what initial conditions to use for the stream.
Flowrate Specification	Specify stream compositions in different forms.
Stream Conditions	Specify temperature and pressure.
Global Data	Specify components and crude cuts and blends.
Import Stream Results	You can also choose to import stream results from an existing DESIGN II Output file.
Stream Specific Crude Data	Specify crude assay property data.

   Referencing Streams:
   You can choose to initialize the stream by using the results from a Reference Stream.

   To add components, click on the Components… button. The Components tab on the Thermodynamic and Transport Methods dialog, displays, allowing you to enter a list of chemical components to be used in your flowsheet model (if you have not already entered them). You can also modify the list. See the ***Specifications*** chapter for details.

   NOTE: Modifying the list of chemical components may affect any component-related data you have already specified.

5. To set time specifications for a stream, click on the **Time Specifications** tab.

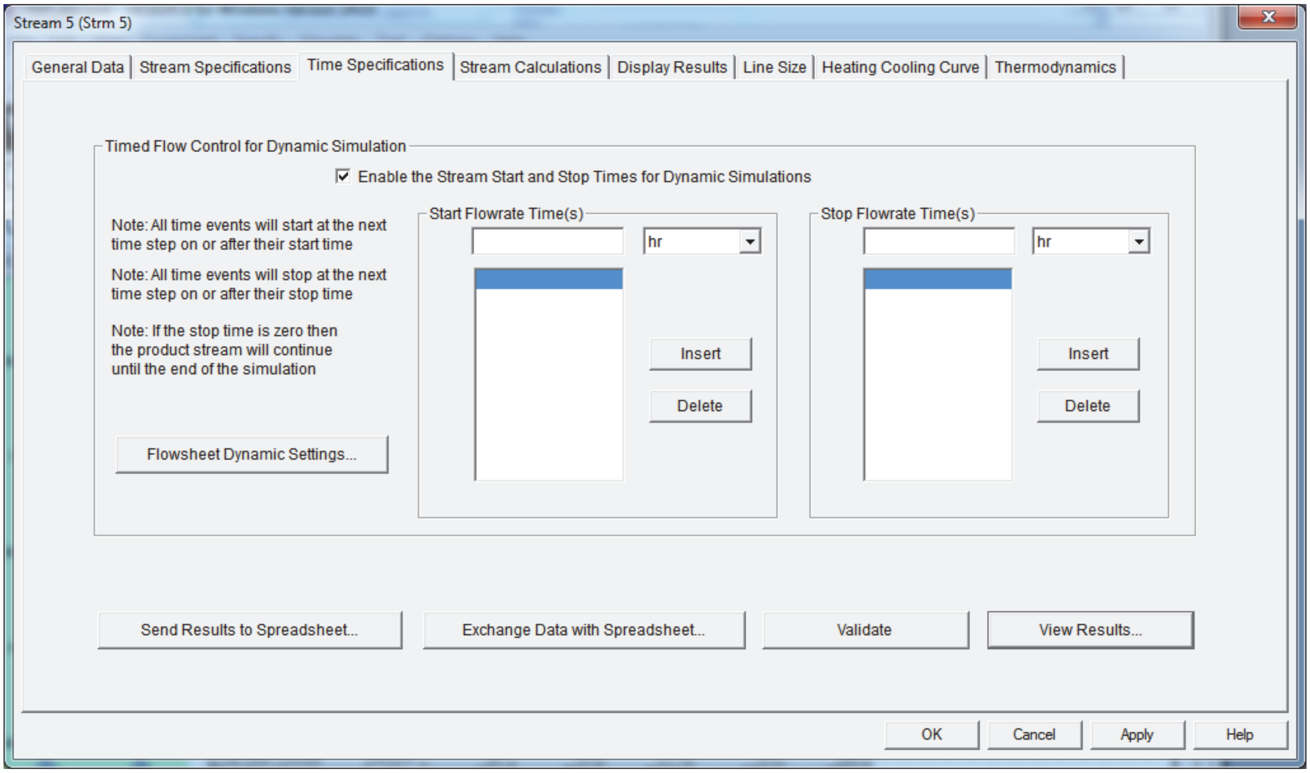


Figure 3: Streams - Time Specifications tab from Stream Dialog (from refi_ex3.psd)

6. Use these settings when running a dynamic simulation; these are not used for Steady State simulations.

Time Step (default 5 minutes)	Enter a time step value and select a unit.
Time Duration (default 60 minutes)	Enter the duration to use and select a unit.

7. To set calculation options for a stream, click on the **Stream Calculations** tab.

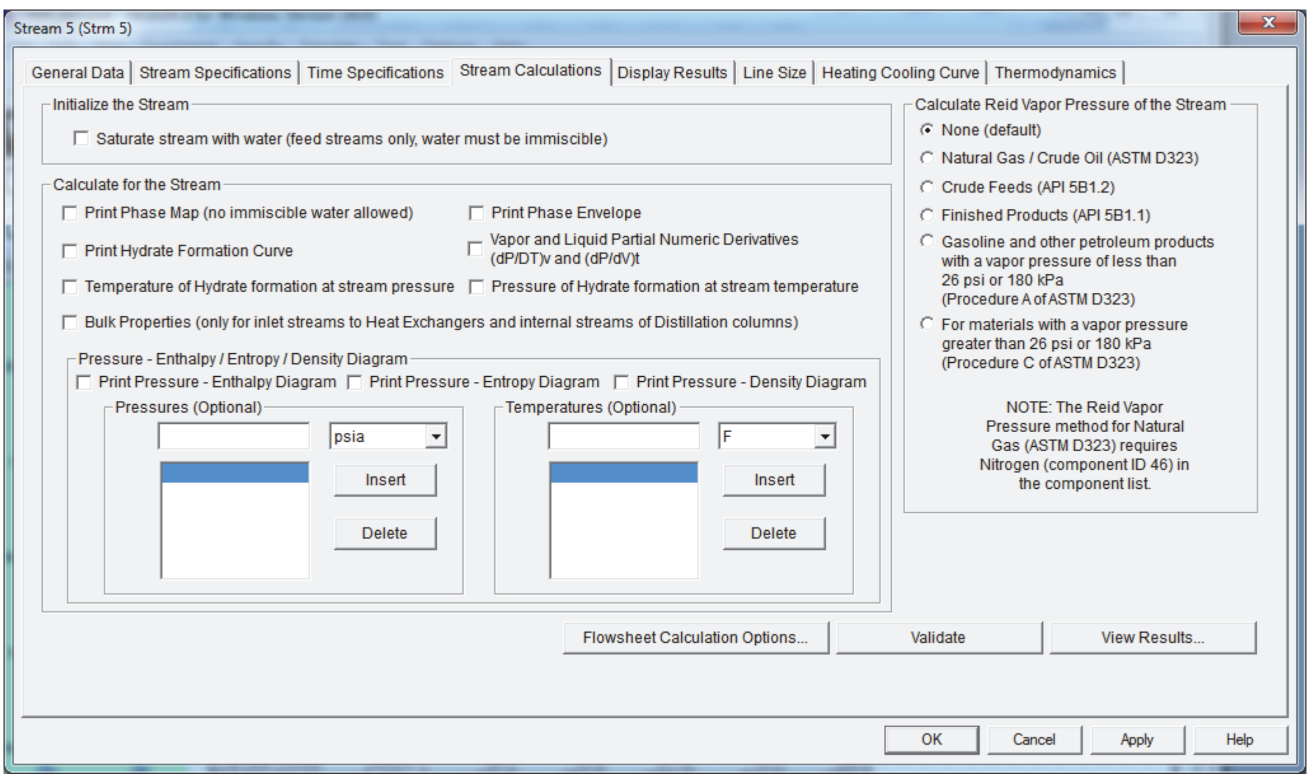


Figure 4: Streams - Calculation Options tab from Stream Dialog (from refi_ex3.psd)

6. Select each category of calculation options (Reid Vapor Pressure, Bulk Properties, Solid CO2 formation, Hydrate formation, Critical Properties and Saturate feed stream with water) on this dialog. You can also select to print a Pressure – Enthalpy diagram and optionally enter pressure and temperature values. If you need help completing this dialog, view the Help topic for the setting stream specifications by clicking the **Help** button.
7. To set heating / cooling curve calculation options for a stream, click on the **Heating Cooling Curve** tab.

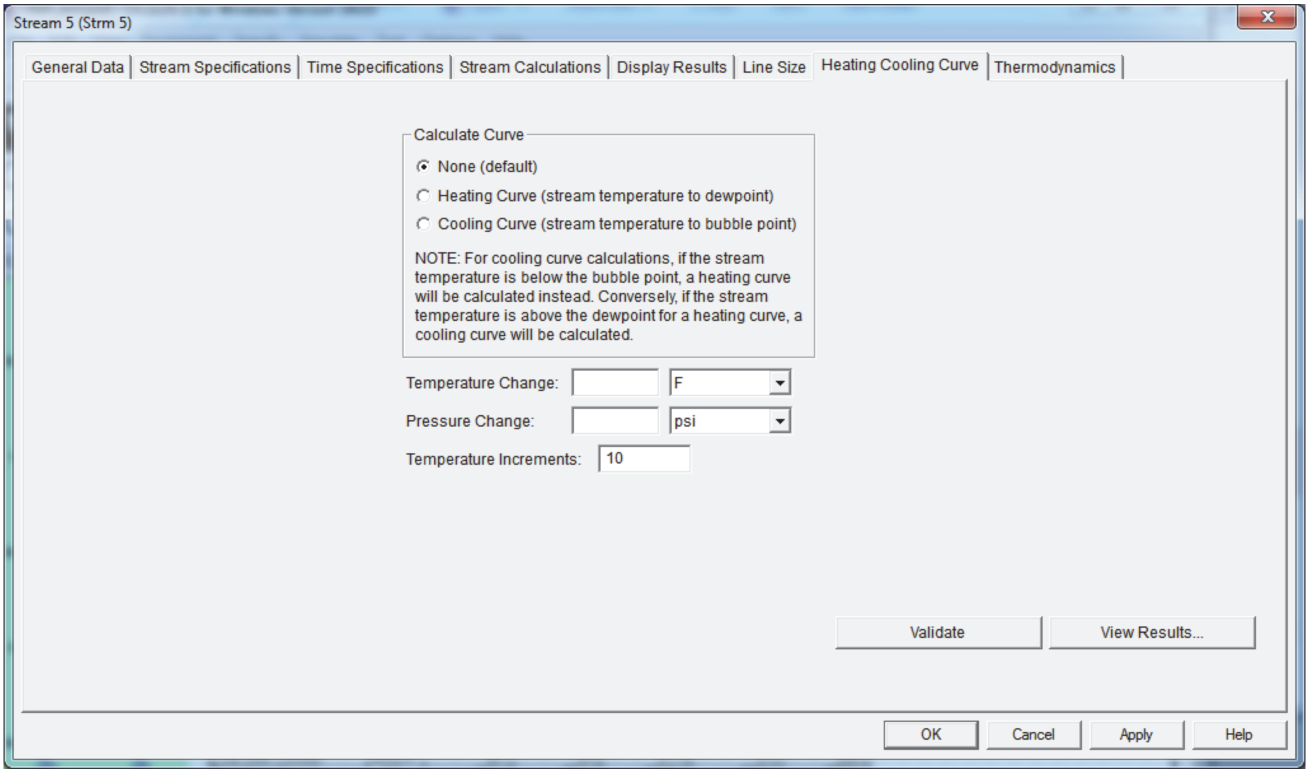


Figure 5: Heating/Cooling Curves tab from Stream Dialog (from refi_ex3.psd)

8. To set Sizing options for a stream, click **on Line Size** tab.

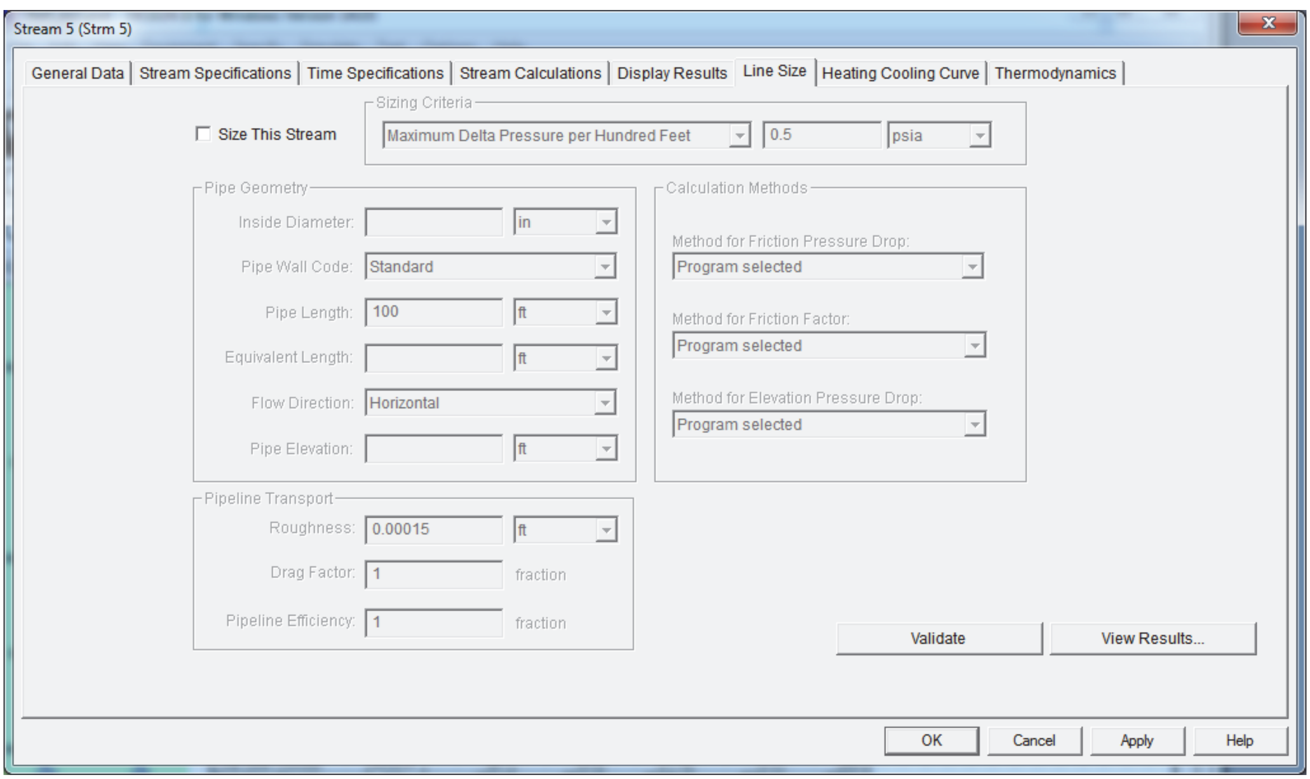


Figure 6: Stream Size Line tab from Strea Dialog (from refi_ex3.psd)

9. To set displayed results for a stream, including results for crude and component flowrates, click on the appropriate **Display** tab (Display Results 1, Display Results 2, Display Results for Crude, Display Component Flowrates 1, or Display Component Flowrates 2.

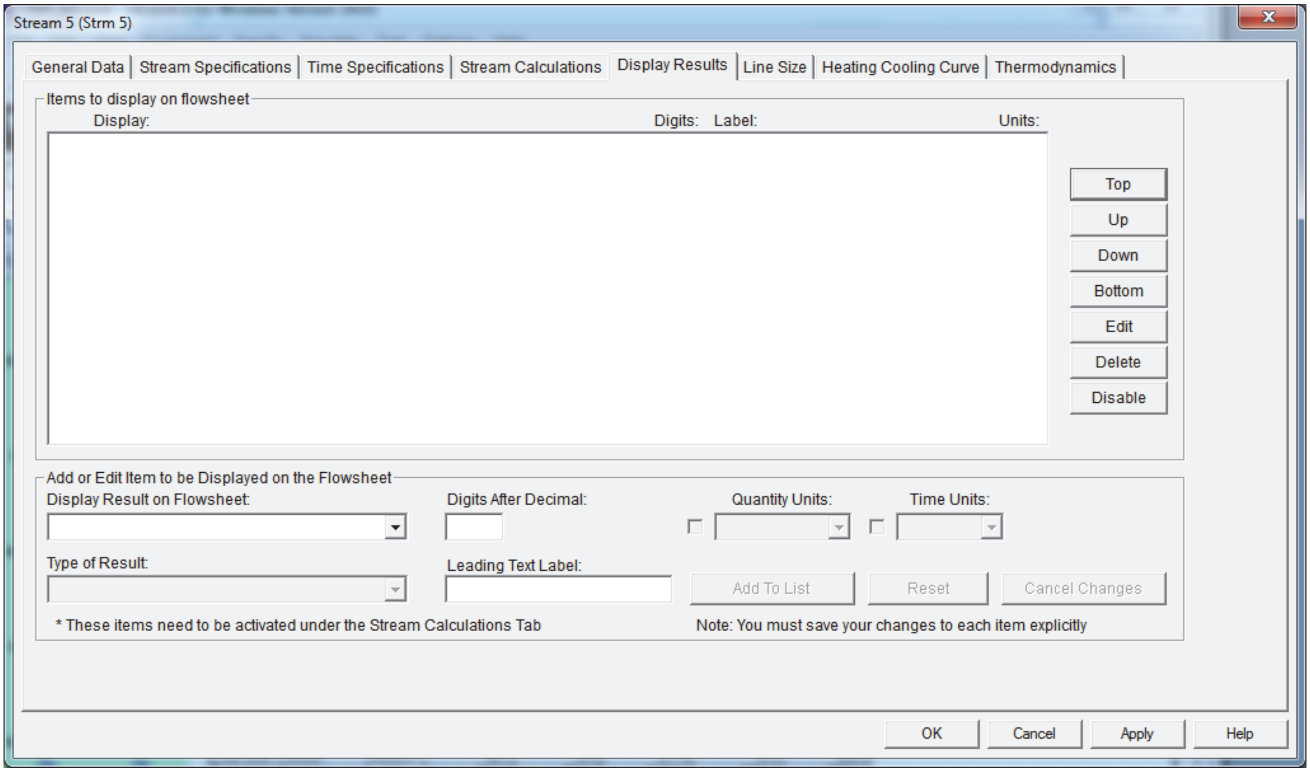


Figure 7: Display Stream Results 1 from Stream Dialog (from refi_ex3.psd)

Petroleum Stream Specifications

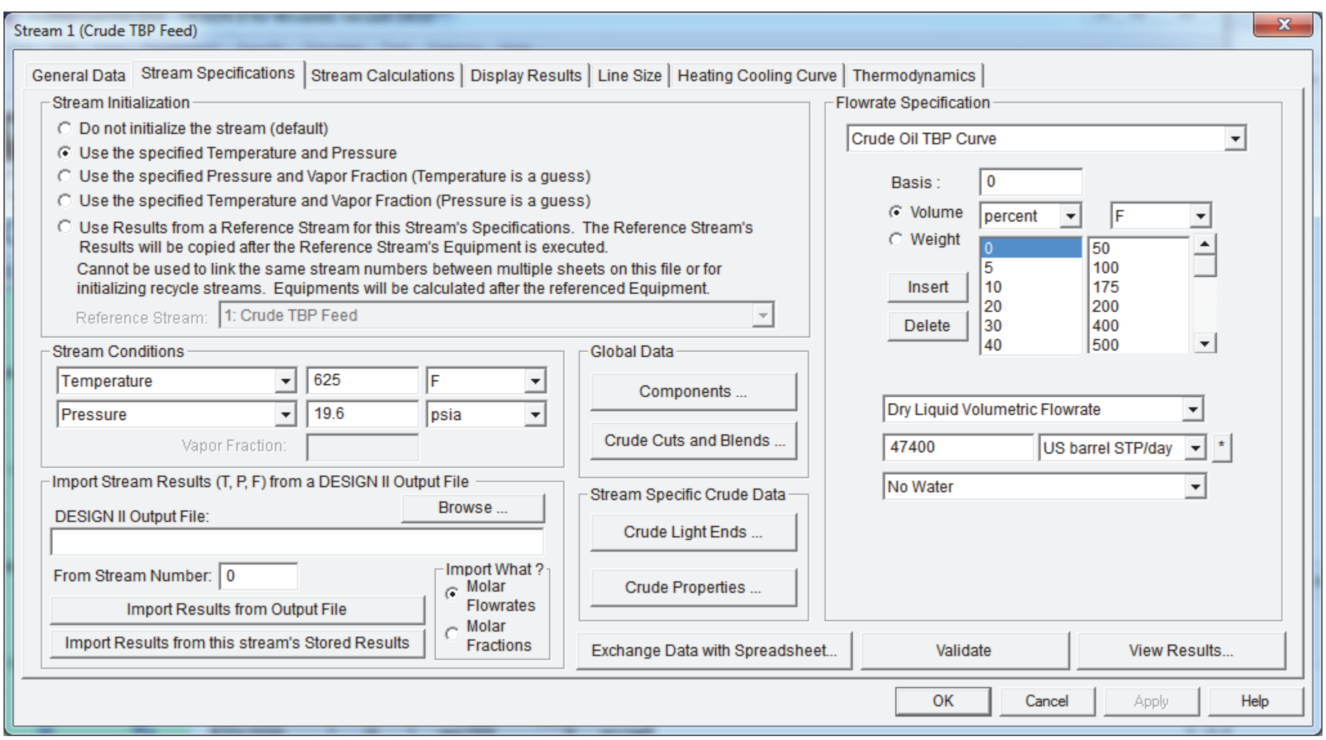


Figure 8: Streams Specifications tab from Stream Dialog (from crudeproperties.psd)

Enter data for each category of entries (Temperature, Pressure and one type of Flowrate specification) on this dialog. If you need help completing this dialog, view the Help topic for the setting stream specifications by clicking the **Help** button.

There are six group boxes for entering stream specifications or data. They are:

Stream Initialization	Specify what initial conditions to use for the stream.
Flowrate Specification	Specify stream compositions in different forms.
Stream Conditions	Specify temperature and pressure.
Global Data	Specify components and crude cuts and blends.
Import Stream Results	You can also choose to import stream results from an existing DESIGN II Output file.
Stream Specific Crude Data	Specify crude assay property data.

To add components, click on the Components … button. The Components tab on the Components Methods dialog, displays, allowing you to enter a list of chemical components to be used in your flowsheet model (if you have not already entered them). You can also modify the list. See the ***Specifications*** chapter for details.

NOTE: Modifying the list of chemical components may affect any component-related data you have already specified.

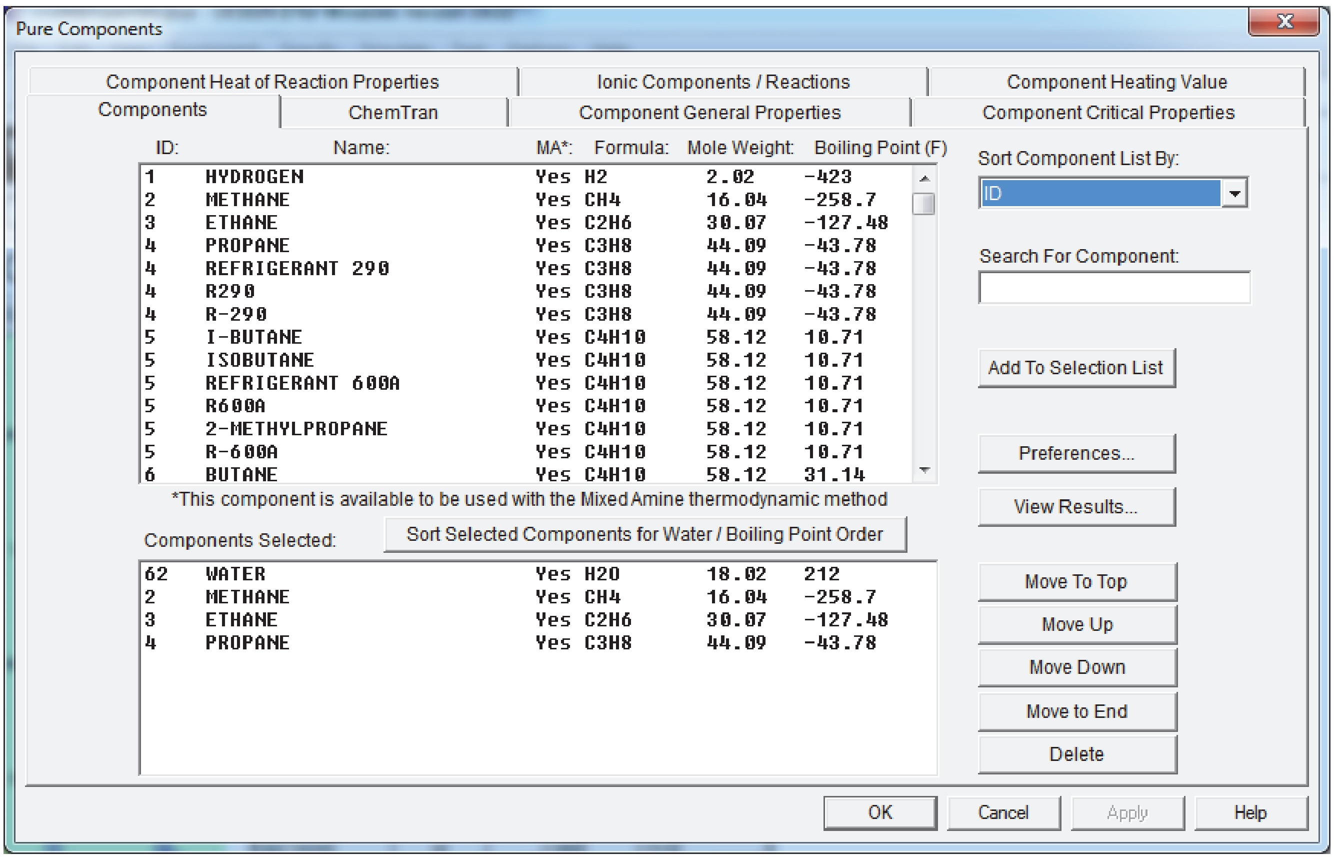


Figure 9: Component Selection tab from Components Methods Dialog (from crudeproperties.psd)

Examining the Pure Component Data Base

The first linked list box describes the chemicals that are in the pure component database. It provides the ID number, assigned name, chemical formula, molecular weight, and the normal boiling point for each of the chemicals. There are several ways of examining the list of chemicals for the ones you want to select:

Sort	The list will appear sorted by ID number. Using the Sort By combo box you can also sort the list by name, mixed amine thermo support, chemical formula, molecular weight or normal boiling point. To perform this operation, select the combo box ; then select the appropriate sort attribute.
Component Search	You can use the Search For edit box to search for a particular chemical component. To perform this operation, enter the Sort By attribute in the Search For edit box. For example, if the sort attribute is ID and you want to locate water with component ID number 62, enter 62 in the Search For box.

Selecting the Chemicals for Flowsheet Simulation

The second linked list box in this dialog lists the components you have selected for your flowsheet model. When you select a chemical it will appear in the bottom linked list box. There are several ways to select chemical components for the model:

Selection with the Mouse	To select chemicals with the mouse, select the chemical from the top linked list box and press the left mouse button twice in quick succession. To select two or more components at a time, hold down the Ctrl key and click on your selections.
Selection with the Add Button	To select chemicals with the Add button, select the chemical from the top linked list box ; then select the Add button.
Deletion of Chemicals	To delete a chemical from the flowsheet model, first select the chemical in the bottom list box ; then select the Delete button.
Scrolling the List	You can examine the contents of the bottom list box by using the scroll bar or the up and down arrow keys on your keyboard.
Changing the Order	To move a component to the top of the list in the list box, first select the component, ; then select the To Top button. Select To Bottom to move the component to the bottom of the list.

To sort the components in the order required for the crude assay data specification, select the Sort For Crude button; this moves water to the beginning of the list and rearranges the rest of the components is ascending boiling point order.

Modifying an existing Chemical Component List

Modify an existing list of chemical components by performing any of the operations described above. If you commit to a different list of components by clicking the OK button, you are warned about component list dependent items that may be affected by the change.

Answer YES on the warning message box to proceed with the change and NO to discontinue.

NOTE: Answering YES will dismiss the Components dialog, while answering NO will leave you at the Components dialog enabling you to either cancel or modify the list further.

If any components are deleted; then any data entered for those components is deleted. If any components are added; then the data for those components is blank.

If the list is rearranged; then the data in component list dependent items is rearranged to correspond to the new list. In most cases you will want to answer YES to the warning.

Crude Cuts and Blends

The Crude Cuts and Blends **. . .** button is found by double clicking on the stream or selecting the stream ; then choosing the **Specify Item** under the **Edit** menu. The Stream dialog displays. The Crude Cuts and Blends button is displayed on the Streem Specifications tab. You can control the crude cuts and/or crude blends to be used in your flowsheet model, if you have not already entered them. You can also modify the list. See the ***Specifications*** chapter for details.

NOTE: Modifying the crude cuts may affect any component-related data you have already specified.

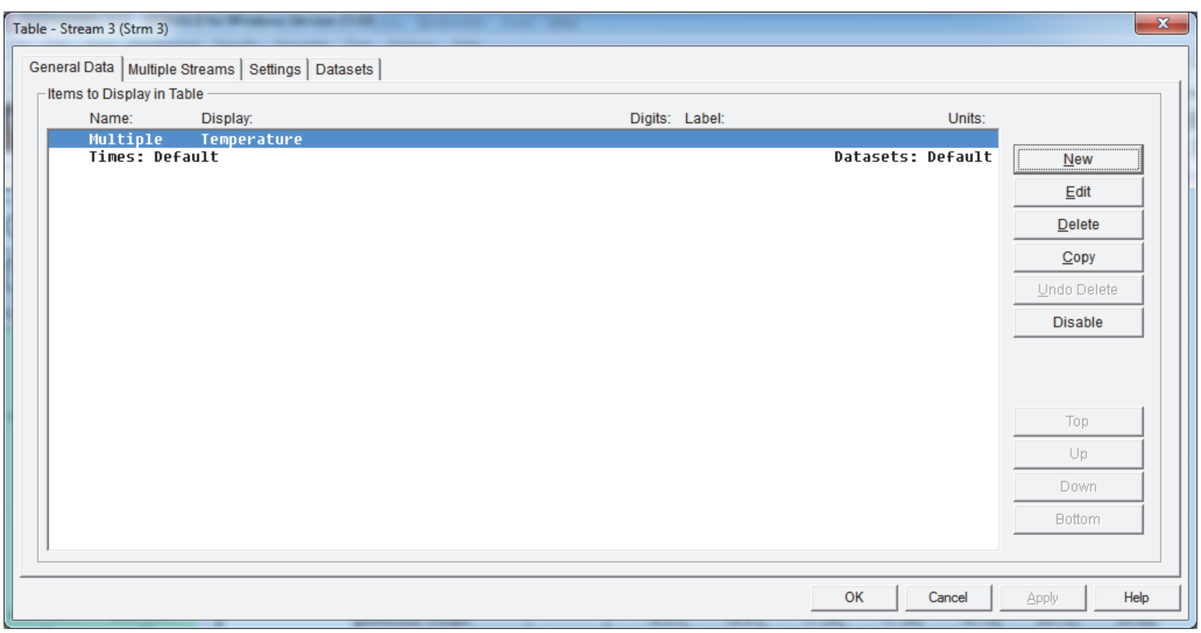


Figure 10: Stream Table Objects: General Data

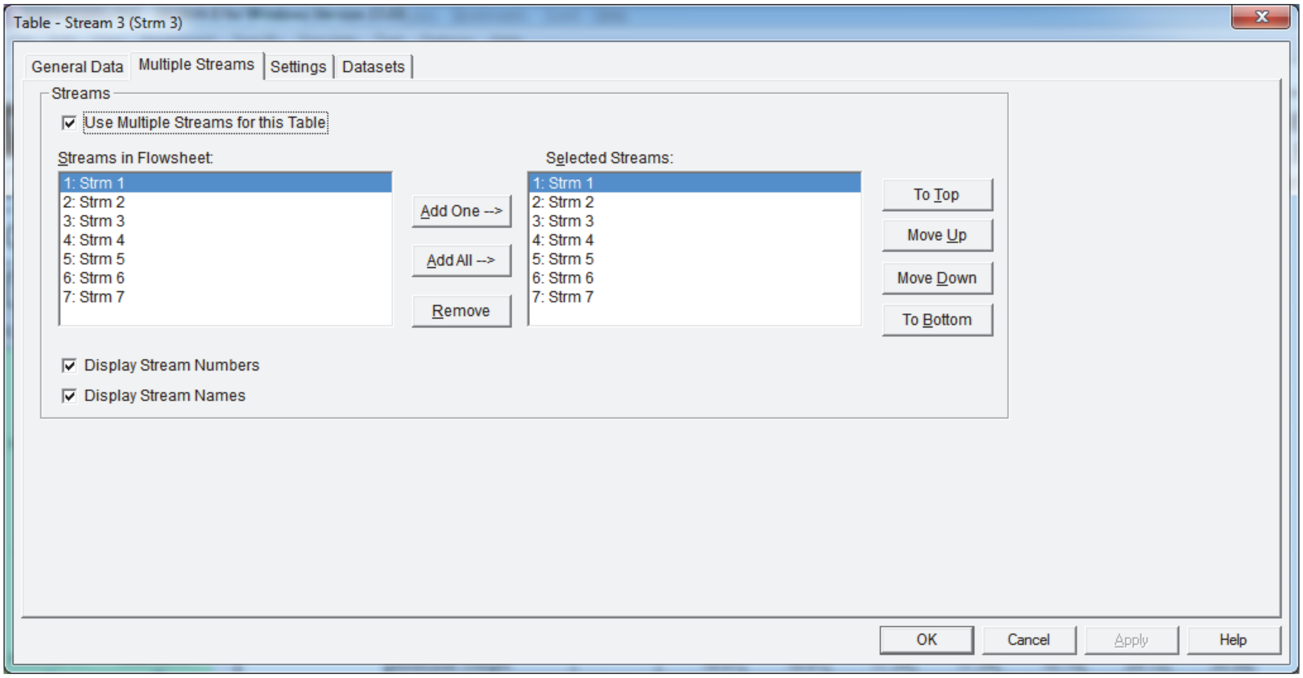


Figure 11: Stream Table Objects: Multiple Streams

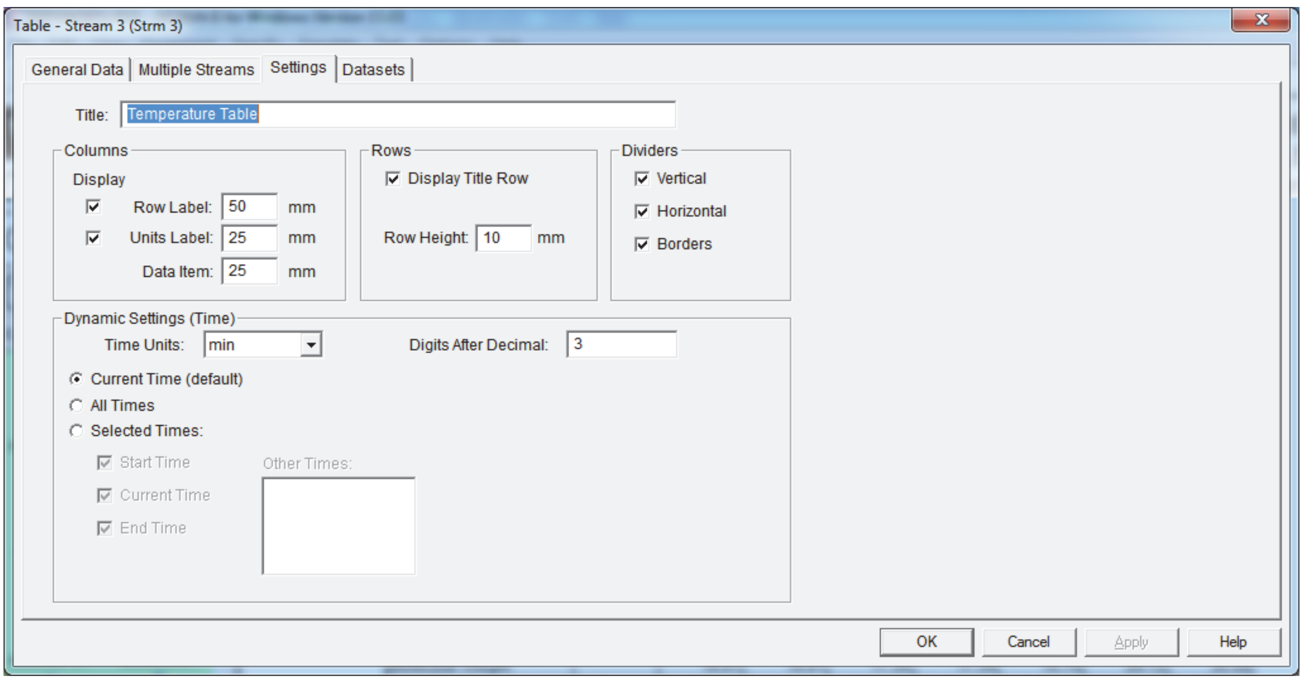


Figure 12: Stream Table Objects: Settings

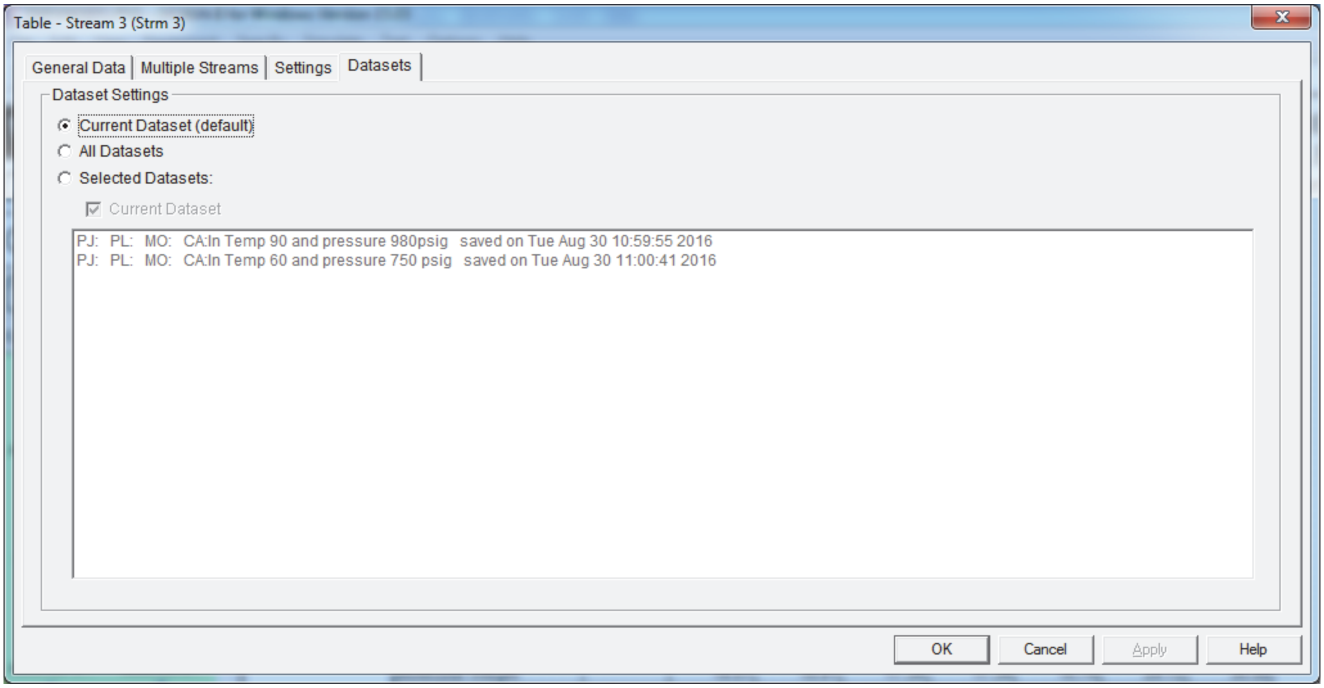


Figure 13: Stream Table Objects: Datasets

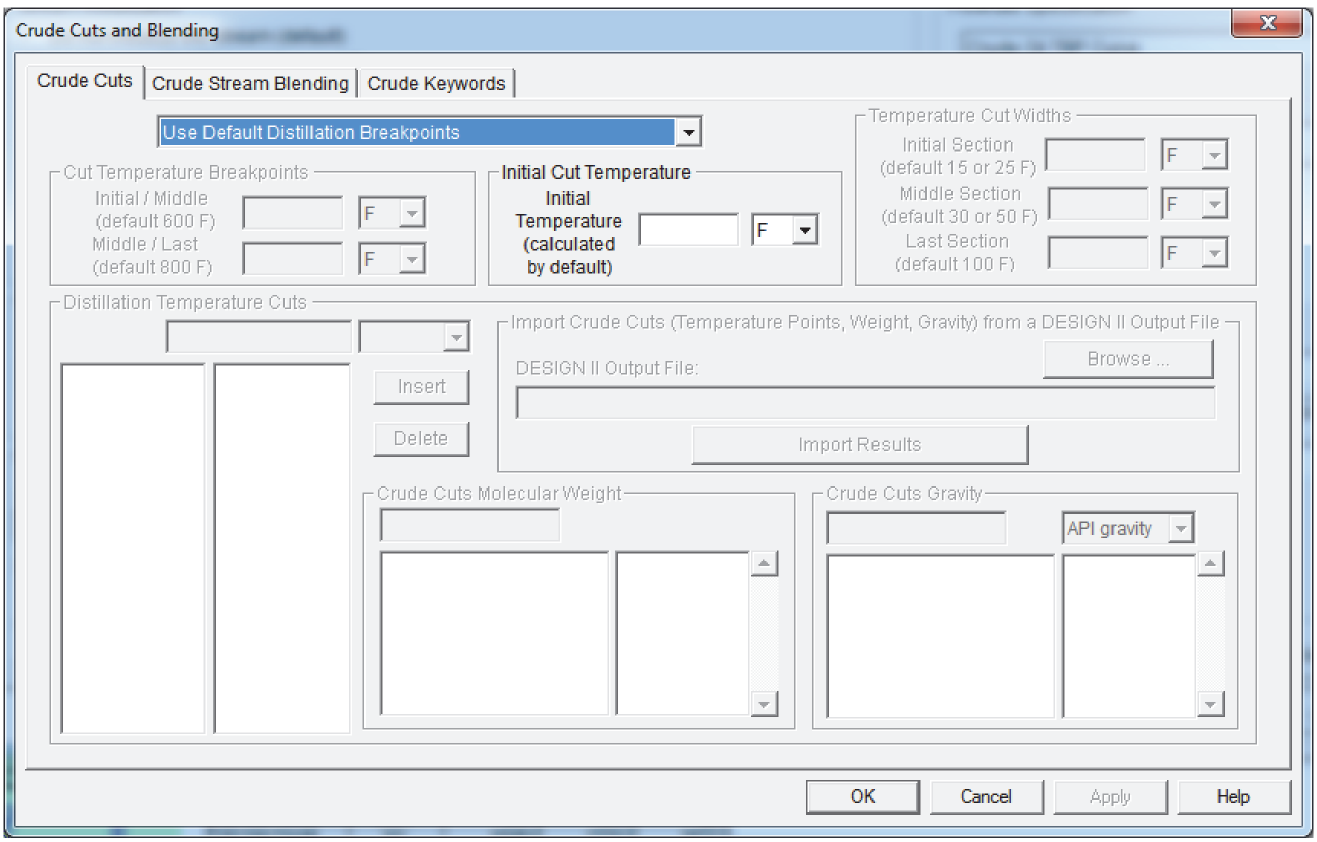


Figure 14: Crude Cuts and Blends tab (from crudeproperties.psd)

Light Ends Analysis

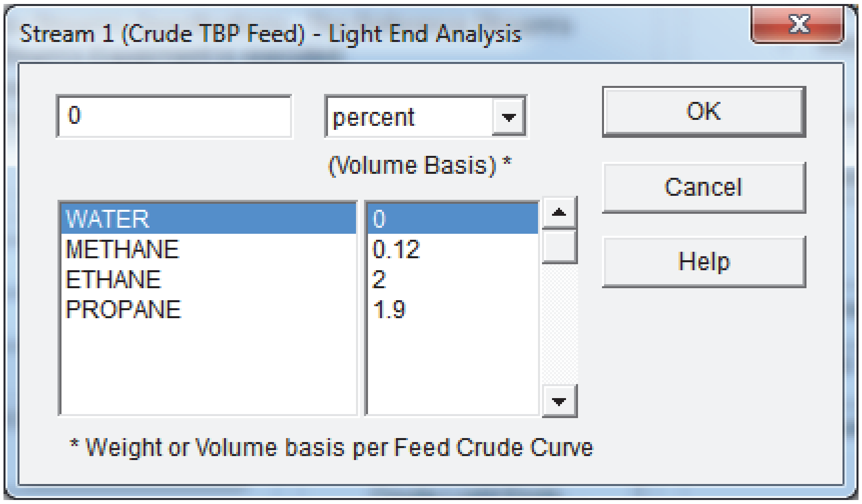


Figure 15: Light End Analysis (from crudeproperties.psd)

Use this dialog to specify the light end portion of a distillation curve, which is specified for the stream in the Flowrate specifications group box of the Basic dialog.

Light ends consist of identifiable chemical compounds (ethane, butanes, etc.) whose properties are contained in the Pure Component Data Base, excluding water (it is handled separately in the Water Flowrate dialog. They can have a major impact on column behavior; always provide this data if you can.

A Linked List Box is provided for entering the light end data. The left column of the table shows the list of components. In the right column of the table, you must enter the liquid volume or weight percent of any components present in the stream. Remember, you chose the volume or weight percent basis in the Flowrate specifications group box of the Basic dialog.

Data Item	Description
Light End Comp Percents	Individual volume (or weight) percents per Feed Crude Curve for components on a dry basis.

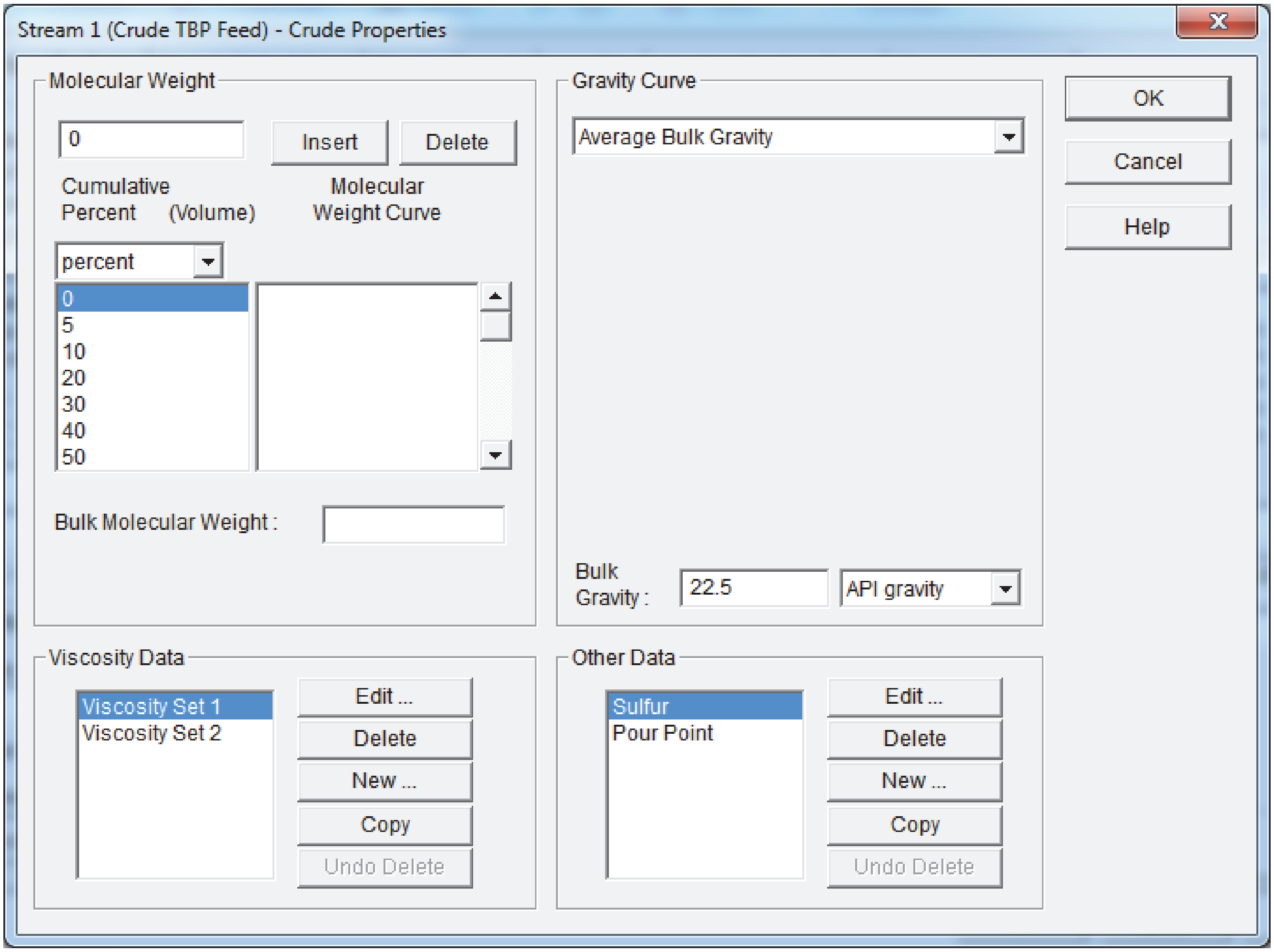


Figure 16: Crude Properties (from crudeproperties.psd)

Crude Properties (Gravity, Molecular Weight, Viscosity and Other Data)

Use this dialog to enter a gravity specification for the stream; it is required along with the distillation curve.

Choose one of three ways to make a gravity specification; make the choice with the combo box.

Enter either an Average Bulk Gravity, cumulative percent distilled versus gravity curve, or Universal Oil Products (UOP) characterization factor.

Enter an Average Bulk Gravity in the edit box and select its units from the adjacent combo box.

Enter a cumulative percent (volume or weight basis) distilled versus gravity curve using the Pair-Table. Remember, you chose the volume or weight percent basis in the Flowrate specifications group box of the Basic dialog.

The cumulative percent column is initially populated with the values entered for the distilled temperatures curve; use the editing functions provided in the Pair Table to override them.

If you enter curve data, you may optionally enter a value for the Bulk Gravity. Though not required, it results in a more realistic curve as the gravities of each pseudocomponent are adjusted to meet the bulk gravity value.

Enter a Universal Oil Products (UOP) characterization factor, which will be used to calculate the gravity of each pseudocomponent.

DESIGN II estimates a molecular weight for each pseudocomponent. You can enter the molecular weights of a feed by entering a molecular weight curve for the stream; do this in conjunction with a distillation curve.

Enter a cumulative percent (volume or weight basis) distilled versus molecular weight curve using the Pair-Table.

Remember, you chose the volume or weight percent basis in the Flowrate specifications group box of the Basic dialog.

The cumulative percent column is initially populated with the values entered for the distilled temperatures curve; use the editing functions provided in the Pair Table to override them.

If you enter curve data, you may optionally enter a value for the Bulk Molecular Weight. Though not required, it results in a more realistic curve as the molecular weights of each pseudocomponent are adjusted to meet the bulk molecular weight value.

Data Item	Description
Average Bulk Gravity	Average bulk gravity for the whole stream.
Gravity Curve	The curve of volume (or weight) percent distilled vs. the gravity.
UOP Characterization Factor	UOP K (or Watson K) factor for calculating the gravity.
Molecular Weight Curve	The curve of volume or weight percent distilled vs. the molecular wt.
Bulk Molecular Weight	A bulk molecular weight for the whole stream.

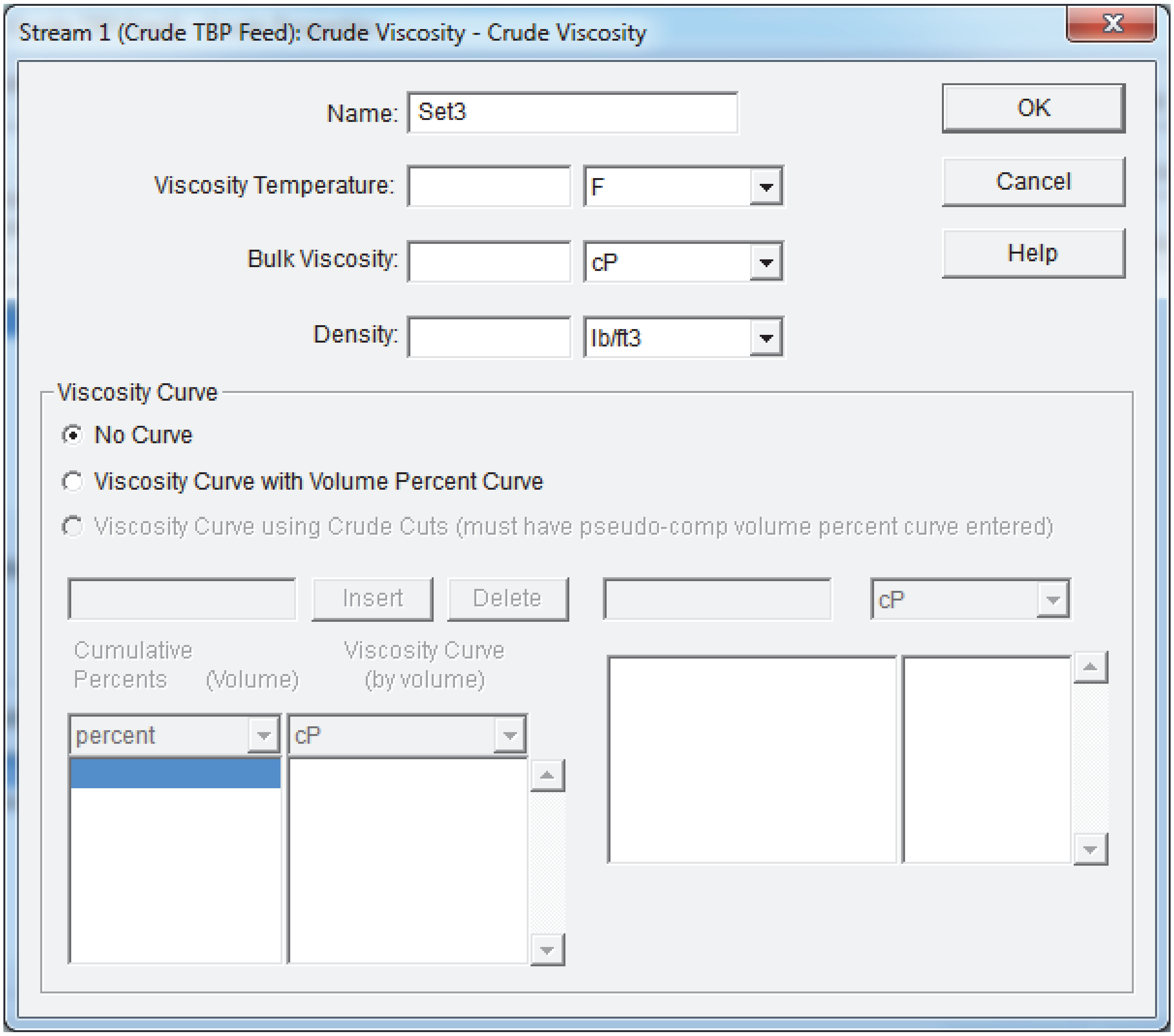


Figure 17: Crude Viscosity Main Dialog (from crudeproperties.psd)

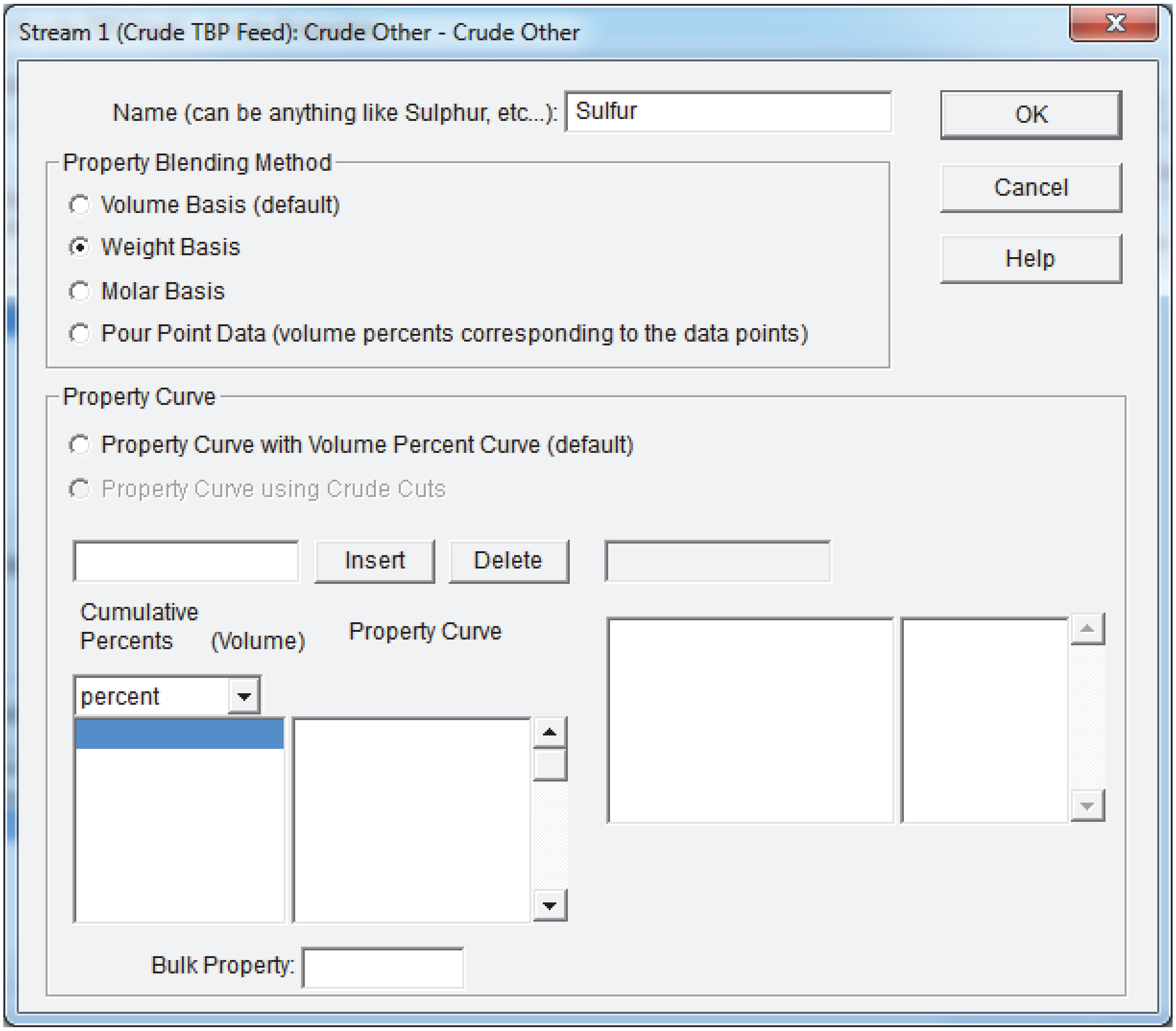


Figure 18: Crude Viscosity Set Dialog (from crudeproperties.psd)

You can enter up to 15 sets of viscosity data for each feed. The viscosity sets are viscosity versus volume percent curve, measured at a specified temperature. If there are multiple feeds, the same number of data sets must be entered for each feed. The viscosity data you enter is used to report a viscosity for each stream and for the REFIne module product specifications. It is NOT used for any equipment sizing calculations such as LINE pressure drop or HEAt EXChanger rating.

In addition to the viscosity data, you should also enter a bulk feed viscosity for each feed.

Feed properties other than viscosity, such as sulfur content, nitrogen content, and pour points, can also be entered as functions of volume percent. Any property you specify, except pour points, must be blendable on a weight, volume or molar basis. In addition, the same number of data sets must be entered for each feed if there are multiple feeds. You can enter a maximum of fifteen sets of data (curves).

If you specify a property, DESIGN II will account for that property in calculating the products. For example, if you specify sulfur content, you will get an idea of sulfur distribution in products.

Property data points are entered using the following command. If your data does not include values for 0 and 100 volume percent, extrapolate your curves and enter values for the initial and end points.

The following restrictions apply:

1. Each feed must have the same number of property and viscosity sets.

2. It is permissible to use FEEd COMponent PROperty SET and FEEd PROperty SET in the same run, as long as only one is entered for a given feed. The same applies to FEEd COMponent VIScosity SET and FEEd VIScosity SET.

3. For any given property set, the same mixing rule option and property name must be used for all feeds.

4. The VIScosity TEMperature command applies to all viscosity sets and should have one value entered for each viscosity set.

Chapter 8: Annotations and Drawing Tools

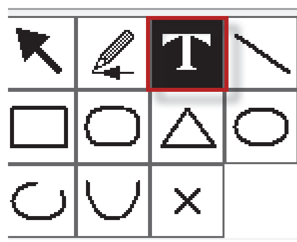
Working With Annotations

You can annotate your flowsheets in DESIGN II for Windows adding text blocks to supplement the text the program automatically assigns to streams and equipment (see the ***Streams*** and ***Equipment*** chapters for details). You can add text any time while creating a flowsheet.

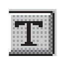
You can move, copy, or delete text blocks. You can also edit text and change text attributes, such as fonts and sizes.

Entering Text

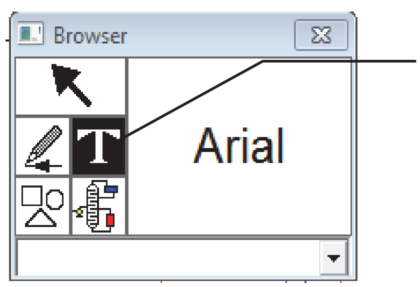
To add text to your flowsheet, you must first click on the **Text** tool of the Toolbox (or optionally the Text tool of the **Browser** if you are using it or the **Text** button on the toolbar).



Text tool on the Toolbox



Text button



Text Tool on the Browser

This places DESIGN II for Windows in its Text mode; all functions you perform in this mode will relate to working with text until you switch to another mode (such as Stream or Equipment mode). The cursor changes to an I-beam when over the drawing area.

You can enter as much text as necessary in a text block—from one character to several lines.

To enter text:

1. Make sure the desired font, style, and size are selected (refer to the ***Changing Text Attributes*** section in this chapter for details).
2. Move the I-beam to the location on the drawing where you want to add text and click the left mouse button. A blinking cursor displays (called the insertion point), which shows you where the text you type will appear (you will still see the I-beam cursor also).
3. Option: to position your text at a different place on the drawing, move the I-beam to that place and click the left mouse button.
4. Begin typing your text.
5. Option: to start a new line, press the Enter key. The cursor moves to the next line and you can continue typing.
6. Option: to delete text, use the backspace or delete key to remove text as necessary.
7. When your text entry is complete, move the I-beam to a blank area on the drawing and click the left mouse button. The text block is complete.

You can also copy text from another source and paste it onto your flowsheet.

You can now either:

- edit the text block you just placed,
- change the text attributes of the text you just typed,

- add more text (repeating steps 2 & 4),
- change to another mode (such as Stream, Equipment, etc.)

Editing Text Blocks

You can move, copy, and delete text blocks, or edit the text in a block.

NOTE: Before editing a text block, make sure DESIGN II for Windows is in the Text mode.

Moving a Text Block

To move a text block:

1. Select the desired text block by placing the I-beam anywhere within the block and clicking the left mouse button. The text block is highlighted.
2. Next place the I-beam over the selected block and press the left mouse button. The I-beam changes to a hand. Drag the block to a new place; then release the mouse button.
3. To deselect a text block, place the I-beam on a blank area of the drawing and click the left mouse button.

Copying a Text Block

You can copy a text block using the **Copy** function under the **Edit** Menu.

To copy a text block:

1. Click on the desired text block; then release the mouse button.
2. Open the **Edit** menu and select **Copy (**to cut the text block and move it to a new place, select **Cut** from the **Edit** menu).
3. Open the **Edit** menu and select **Paste**. A copy of the text block is placed in the middle of the screen, while the original text block remains in place.

Deleting a Text Block

To delete a text block:

1. Place the I-beam cursor anywhere within the block and click the left mouse button. The text block is highlighted.
2. Click the right mouse button.
   NOTE: the text block is removed immediately; no warning message appears.

Editing Text

To edit text:

1. Place the I-beam cursor anywhere within the block and double-click the left mouse button. The text block is highlighted, and the insertion point appears in the text block.
2. Make any changes or deletions.
3. To deselect a text block, place the I-beam on a blank area of the drawing and click the left mouse button.

Changing Text Attributes (fonts & sizes)

You can change text attributes such as font, size, and style.

NOTE: any attributes you set are used as the default until you set new ones or exit from DESIGN II for Windows. If you have the Auto-Save option on, the attributes will remain in effect.

To change text attributes:

1. Select the desired text block by placing the I-beam anywhere within the block and clicking the left mouse button. The text block is highlighted.
2. Select the desired text attributes using one of the options below:

Setting Font Size

Option 1: to change the size using a pre-set size description (tiny, medium, etc.), open the Text menu and select one of the following:

Following are the available sizes and their measurement:

Tiny Font size is 6 points.
Small Font size is 11 points.
Medium Font size is 23 points.
Large Font size is 34 points.
Huge Font size is 45 points.

Option 2: to change the size using a point size, open the **Text** menu and select **Fonts...** , or click on the **Font** button on the toolbar (you can also hold down the **Ctrl** key and press the letter **T**). The Fonts dialog displays.

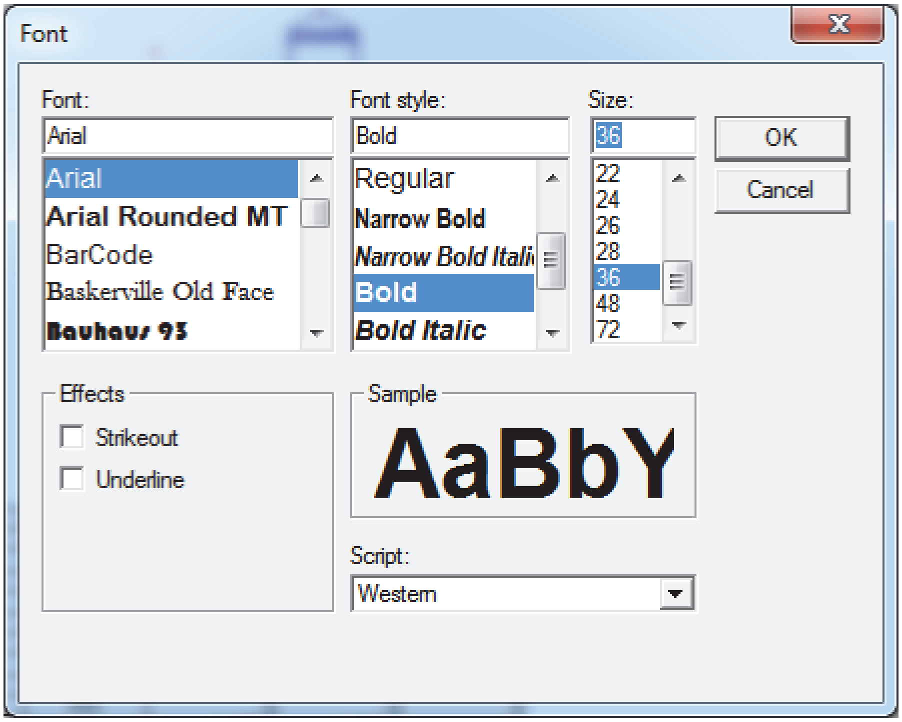


Figure 1: Font Dialog

Type in a point size (72 points = 1 inch) in the **Size** field or scroll through the size list and click on the desired size. The Sample field on the dialog shows how the text will look. Click **OK** to apply the change to the text.

Setting Font Type and Style

1. Option: to change the font for the text block, open the **Text** menu and select **Fonts...** , or click on the **Font** button on the toolbar (you can also hold down the **Ctrl** key and press the letter **T**). The Fonts dialog displays. Scroll through the font list; then click on the desired one. The Sample field on the dialog shows how the text will look. Click **OK** to apply the change to the text.
2. Option: to change the font style (bold, italic, etc.), open the **Text** menu and select **Fonts...** , or click on the **Font** button on the toolbar (you can also hold down the **Ctrl** key and press the letter **T**). The Fonts dialog displays. Click on the desired style in the Font Style list, or click the **checkbox** next to Strikeout or Underline. The Sample field on the dialog shows how the text will look. Click **OK** to apply the change to the text.

Hiding Text

You may want to “hide” text on a drawing, or not display it, without actually deleting the text. You can later “show”, or display, any hidden text.

To hide text:

1. Open the **Text** Menu and choose **Hide Text...** . The Hide Text dialog displays.

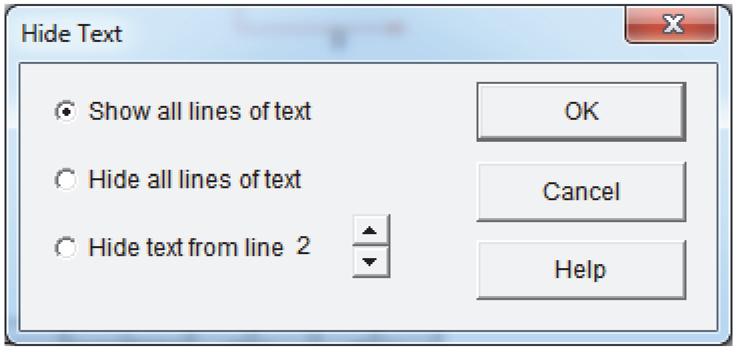


Figure 2: Hide Text Dialog

2. Click on the **button** next to your desired choice, either **Hide text from line x** (where x is the line number) or **Hide all lines of text**. If you select **Hide text from line x**, click on the up or down arrow to increase/decrease the number of lines to hide.
3. Click **OK**. The selected amount of text is hidden for all text blocks on the drawing.

To show text:

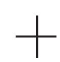
1. Open the **Text** Menu and choose **Hide Text...** . The Hide Text dialog displays.
2. Click on the **button** next to your desired choice, **Show all lines of text**.
3. Click **OK**. All hidden text is displayed for all text blocks on the drawing

Working With Drawing Tools

To add drawing elements (e.g. line, rectangle, arc) to your flowsheet you must first place the Browser in drawing element mode. You can select a drawing element for placement on the flowsheet from the Browser list box.

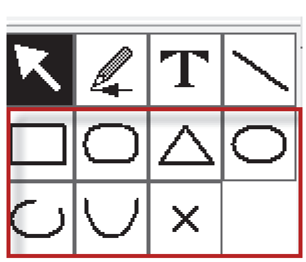
You can choose to add:

- Lines
- Rectangles
- Rounded Rectangles
- Triangles
- Ellipses
- Arcs
- Parabolas
- Snap points

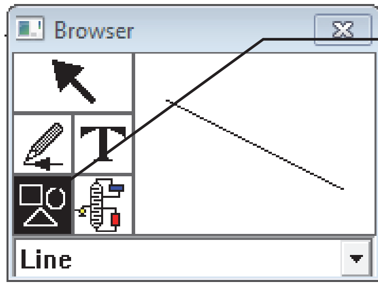
When the Browser is in drawing element mode and the cursor is in the drawing area, it will appear as a  with the selected drawing element next to it.

Adding Drawing Elements

To add drawing elements (e.g. line, rectangle, arc) to your flowsheet you must first select the desired drawing tool from the Toolbox (option: place the Browser in drawing element mode and selecting a drawing element for placement on the flowsheet from the Browser list box).



Drawing tools on the Toolbox



Drawing Elements Tool on the Browser

Selecting Drawing Elements

Make sure the Browser is in the selection mode, ; then click on the element. The element is highlighted and handles appear on it.

Moving Drawing Elements

Once the element is selected, you can move it by placing the cursor anywhere in the element, pressing the left mouse button, and holding it down. The  will appear. This signals to you that you can move the element by moving the mouse. Release the mouse when you have repositioned the element. You de-select a element by placing the cursor outside the element and pressing the left mouse button.

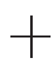
Resizing Drawing Elements

With the element selected, click on a handle and drag it to a new size.

Copying Drawing Elements

You can copy an element with the **Copy** function under the **Edit Menu**. First, select a region enclosing the element. Next, select the **Copy** option. Finally, use the **Paste** function.

Deleting Drawing Elements

To delete an element, place the cursor  anywhere within the element and press the right mouse button. Select Delete from the pop up menu.

Creating Custom Symbols

You can use the primitive drawing objects to create symbols for use in a flowsheet.

Creating Custom Symbol Libraries

You can turn a group of selected graphical primitives into a new symbol, which can then be added to a new or existing custom symbol library.

After drawing the primitives on your flowsheet, drag open a selection box to include all the objects you have drawn. You can create snap points for the symbol or let DESIGN II for Windows place them. You can select Snap points from the list of drawing elements.

Open the File menu and select Create Symbol from Selected Graphics to display this dialog.

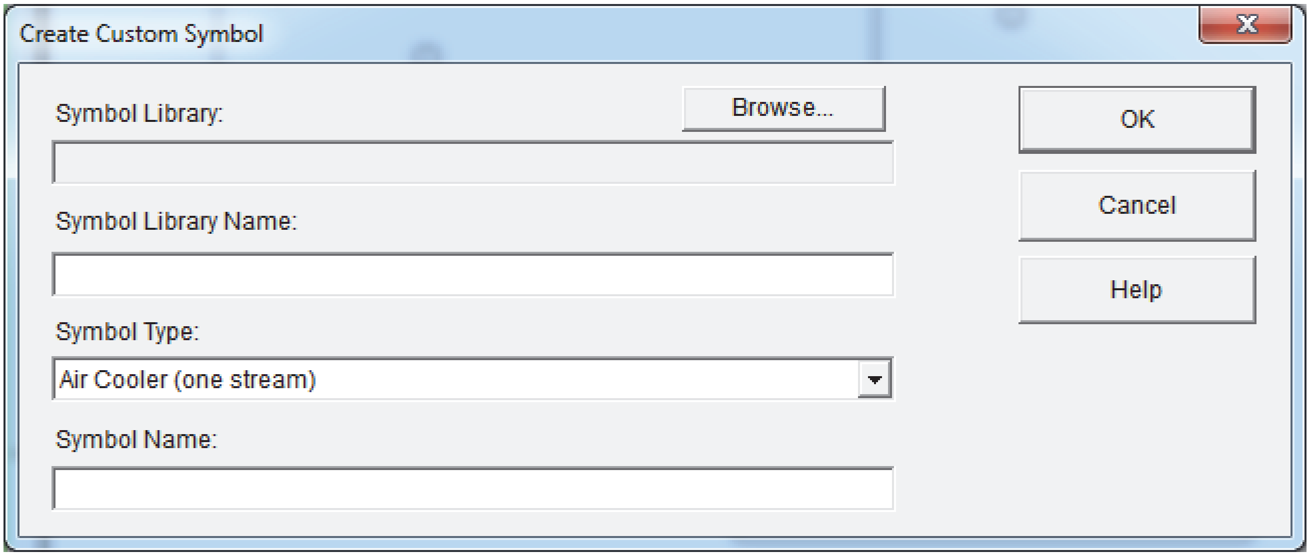


Figure 3: Create Custom Symbol Dialog

Following are the options on the Create Custom Symbol dialog.

Symbol Library The field shows the location of the symbol library file. You can browse to an existing library, or create a new one. The symbols library file has the extension .sym.

Symbol Library Name Enter a name for a new library. For existing libraries, this displays the library name, and cannot be changed.

Symbol Type Open the list and choose which equipment type you want this new symbol to be.

Symbol Name Enter the name for the new symbol.

Managing Custom Symbol Libraries

You can create custom symbols for use in your flowsheet. You can either add the symbol to a new library or existing library. Each library you create is added to the Equipment menu and Browser equipment list box under the library name.

You will use the Manage Custom Symbol Libraries to create new libraries or remove existing ones. You can view the names of the symbols in a library and delete any symbol from the library.

You can also choose which, if any, standard Design II libraries you would like loaded.

Open the File menu and select Manage Custom Symbol Libraries.

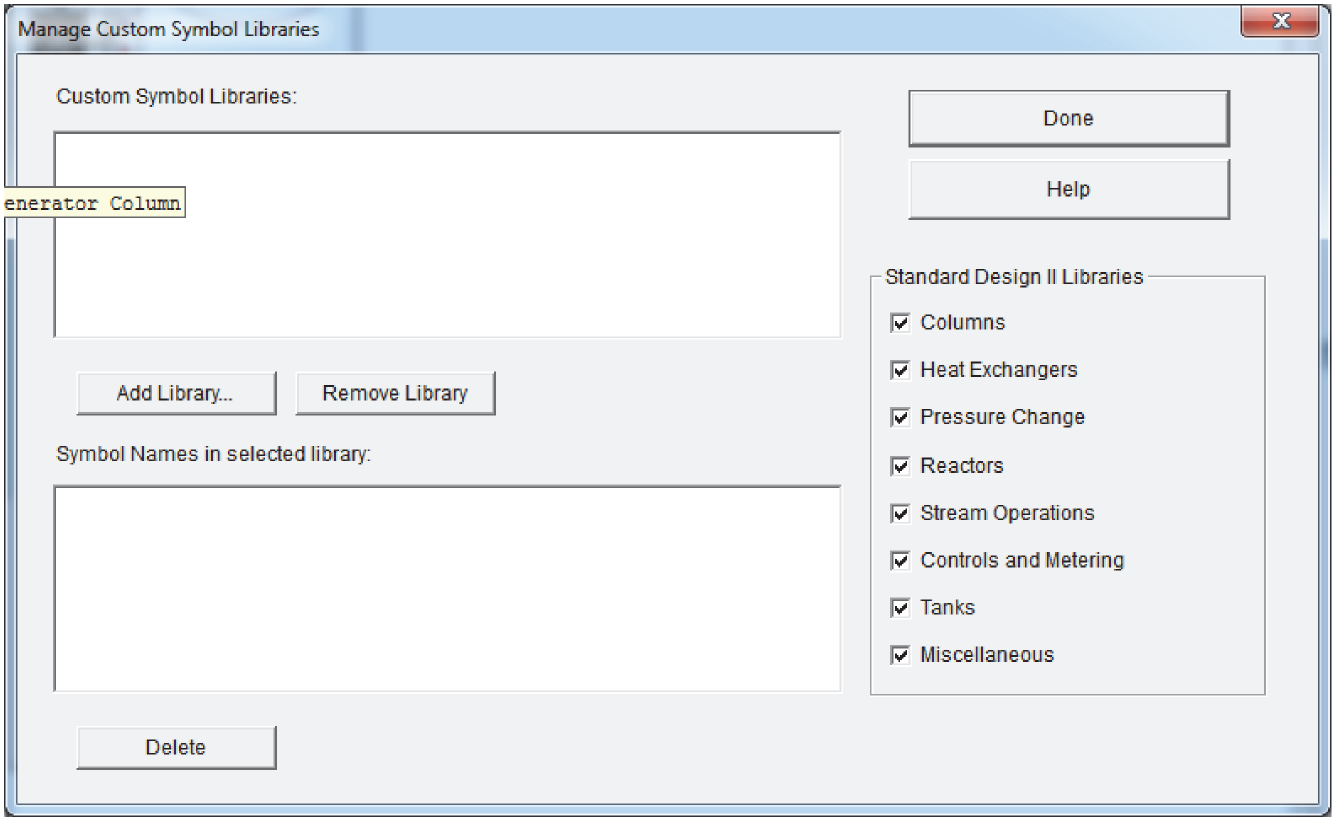


Figure 4: Manage Custom Symbol Libraries Dialog

Custom Symbol Libraries — A list of custom symbol libraries currently loaded and available.

Add Library — Click this button to browse for an existing user created library, which will then be added to the list.

Remove Library — This button removes the currently selected library from the list, but this does NOT delete the library.

Symbol names in selected library — When a library is selected in the above list, the symbols in that library are listed here by name.

Delete — To remove a symbol from the list of the selected library, click this button. The selected symbol is removed from the corresponding library and CANNOT be undone.

Standard Design II Libraries — Choose which standard libraries you would like to use.

Using Pop Up Menu Functions

You can also edit an element by right clicking on it. Along with cutting/copying/pasting/deleting it, you can change the color and fill color (for certain element), rotate it at an angle (90 or 180 degrees), flip it horizontally and vertically, and change the line style (solid, dash, dot, dash dot, dash dot dot, and thick). You can also bring the element to the top layer of the drawing, if it is behind another object in the drawing area.

Adding Bitmap Graphics

You can also paste a graphic from another source onto the flowsheet. You must create a graphic using a third party graphics program and then copy it using the Edit/Copy function.

An option is to locate an existing graphic from another source and make sure it is copied onto your PC. You can then open the graphic in a third party graphic program; then select the Copy function. Refer to the third party graphics program Documentation/help for assistance.

Once the graphic is copied, you can move to DESIGN II for Windows and open the Edit menu; then select Paste. An option is to make sure the pointer cursor is selected from the browser tool; then right-click on the menu and select Paste. The graphic is pasted next to where the cursor is located on the flowsheet. See the Help file for more details on working with graphics.

Chapter 9: General Specifications

Components

You can select the list of chemical components for your flowsheet model.

To select these components:

1. Open the **Specify** menu and select **Components**. The Components dialog displays, with the Components tab selected. The top list box displays all available chemical components. The bottom list box (empty at first) will display all selected components.

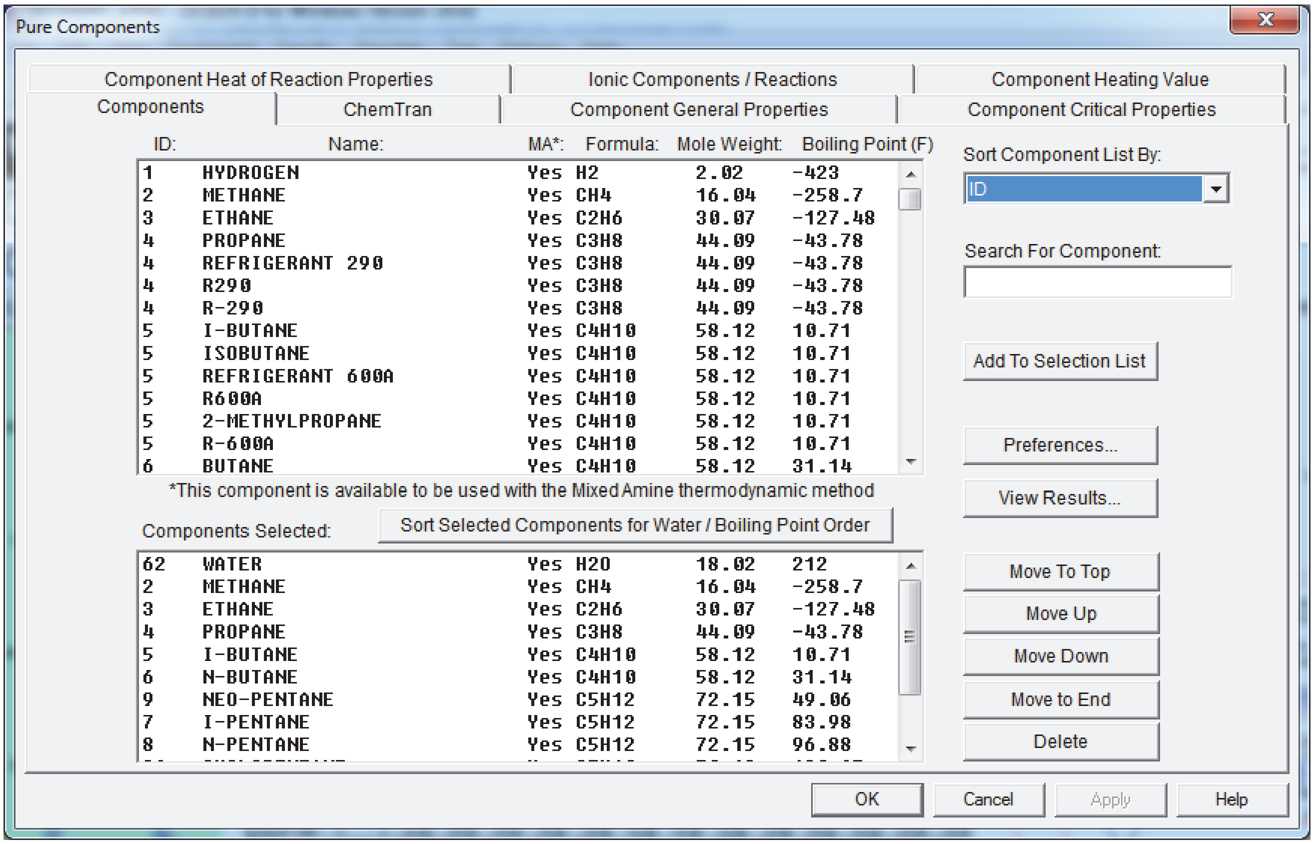


Figure 1: Components tab (from refinery.psd)

2. Select the desired components. Refer to the following sections for details: ***Sorting the chemical component list/searching for a chemical component***; ***Selecting a chemical component***; ***Changing the order of selected components***; and ***Deleting a selected component***.

3. Click the **OK** button when done. The Component Selection dialog closes.

Sorting the Chemical Component List / Searching for a Chemical Component

The first linked list box on this dialog displays the chemicals in the Pure Component database. It lists the ID number, assigned name, chemical formula, molecular weight, and the normal boiling point for each chemical. You can sort this list or search for a desired chemical

Sort

The chemical list is initially sorted by ID number. You can also sort the list by name, chemical formula, molecular weight, or normal boiling point. Open the drop down **Sort** list ; then select the appropriate sort attribute.

Search For

You can use the **Search For** field to search for a particular chemical component. First, make sure the list is sorted the way you want it. Then, enter the **Sort By** attribute in the **Search For** field. For example, if the sort attribute is ID and you want to locate water with component ID number 62, enter *62* in the **Search For** box. The list is sorted as you type, until the matching chemical is displayed and highlighted.

Scroll

You can scroll through the list of components using the **scroll bar** to the right of the list.

Selecting a Chemical Component
With the desired component displayed, either double-click on the chemical component name, or click on the name and click the **Add** button. To select two or more components, hold down the Ctrl key and click on the desired components. The chemical component is added to the bottom list box of selected components.

Changing the Order of Selected Components
To move a component in the list box:

1. Click on the component name; you may need to scroll to display the name.
2. Then, click on the **To Top** or **To End** button. The component is moved to the desired position.
   To sort the components into the order required for the crude assay data specification, click on the **Sort For Crudes** button; this moves water to the beginning of the list and rearranges the rest of the components in ascending boiling point order.

Deleting a Selected Component
To delete a selected component:

1. Click on the component name in the bottom list box; you may need to scroll to display the name.
2. Click the **Delete** button; the component is removed from the list of selected components, not from the pure component database.
   NOTE: deleting a component will also remove any associated data for the component in streams that have been specified.

Thermodynamic Methods

You may select thermophysical property options for the flowsheet. If you do not, a default equation of state methods will be used. Follow these steps:

1. Open the **Specify** menu and select **Basic thermo**. The Thermodynamic and Transport Methods dialog displays with the Thermodynamic Methods tab selected.

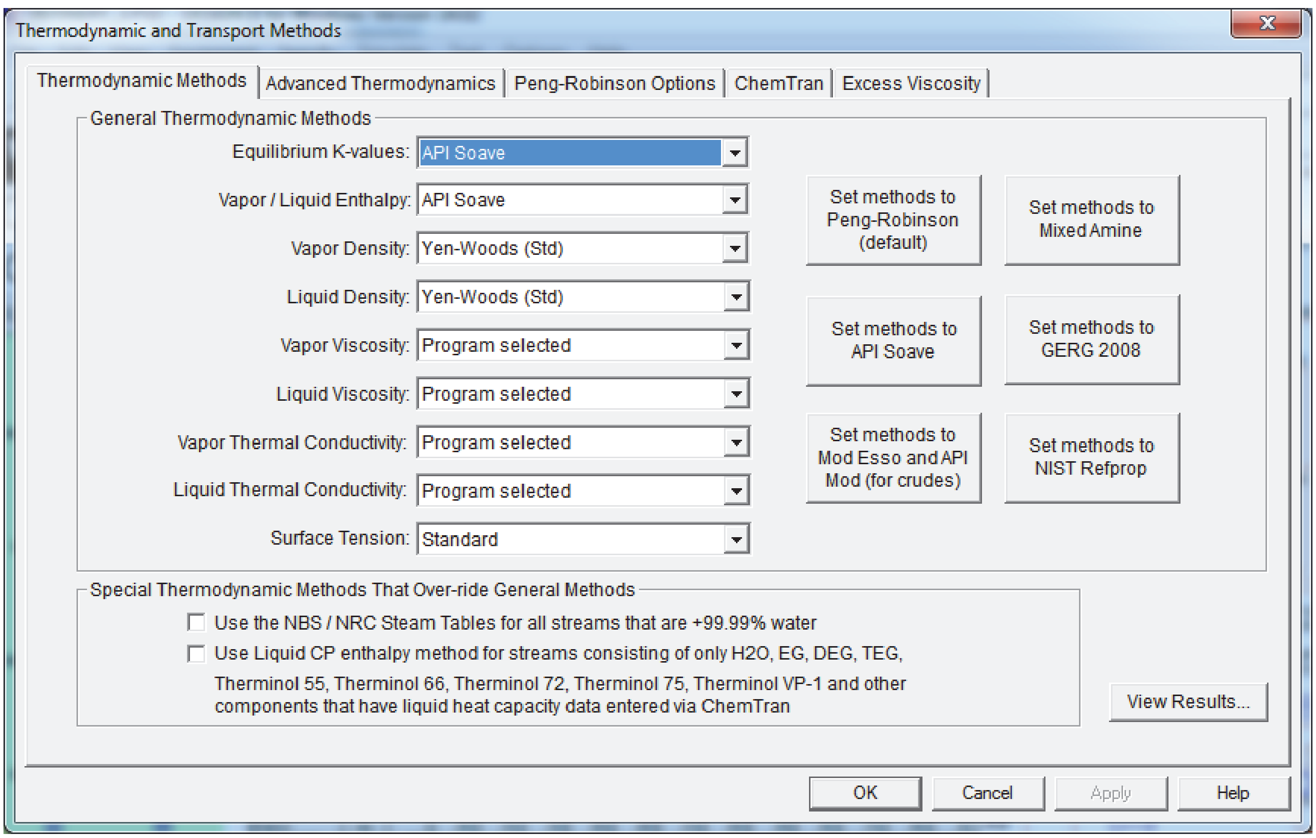


Figure 2: Thermodynamic Methods tab

2. Set the desired correlations for thermophysical properties. Open the **drop down** list next to the desired calculation type and select the desired method.

   Equilibrium K-Values (API Soave is the default)

Vapor/Liquid Enthalpy (API Soave is the default)
Vapor/Liquid Density (Yen-Woods Std is the default)
Vapor/Liquid Viscosity (Program Selected is the default)
Vapor/Liquid Thermal Conductivity (Program Selected is the default)
Surface Tension (Standard is the default)

NOTE: For viscosity and thermal conductivity calculations, Program Selected means standard DESIGN II defaults are used. Check your online ***DESIGN II*** Reference Help topic "Thermodynamics" or the DESIGN II Reference Guide "Chapter 5:Thermodynamics" for a description of these defaults.

Methods are changed for input files with Crude feed sections. Check your online ***DESIGN II User Guide*** for details on these methods.

3. Click the **OK** button when done. The desired methods are set.

Advanced Thermodynamics

Use Advanced Thermodynamics to specify a chemical file name and select water calculations.

1. Open the **Specify** menu and select **Basic thermo**. The Thermodynamic and Transport Methods dialog displays with the Thermodynamic Methods tab selected. Click on the **Advanced Thermodynamics** tab.

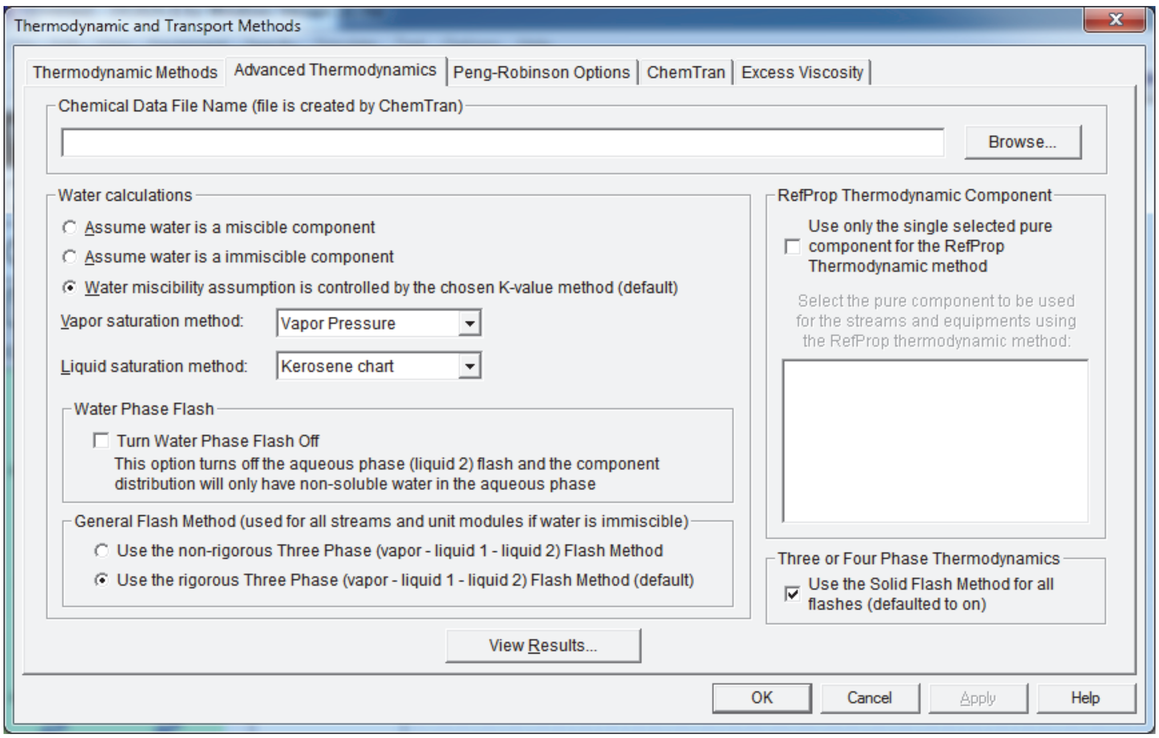


Figure 3: Advanced Thermodynamics tab

2. Select the desired option you want to use and complete the necessary information.

Chemical File

For some chemical systems, you may need to provide additional thermophysical property data to DESIGN II to model your flowsheet. You can use WinSim's thermophysical property estimation and correlation program, ChemTran, to create this data. This data is transferred from ChemTran to DESIGN II through a binary file. Use the edit boxes in this group box to enter the required name for the chemical file.

Water Calculations

DESIGN II provides an option to treat water in a special way for mixtures containing oil and water at conditions where two separate liquid phases definitely form. This option specifies water is a separate, pure liquid phase when chemicals are distributed between the vapor and liquid oil phases based on the thermodynamic equilibrium model.

You have options for specifying how much water is in the liquid oil phase based on correlations for solubilities of water in oil. You also have options for specifying how DESIGN II determines vapor phase saturation with water.

You have three radio buttons for specifying the assumptions made about water distribution among phases. Combo boxes are provided for you to choose the option to calculate the amount of water in the vapor and liquid hydrocarbon phases.

Immiscibility/Radio Buttons

Check one of the following:

Never assume water is immiscible

For certain K-value options, the assumption that water is immiscible is turned off automatically. If you select this button, DESIGN II will calculate the distribution of water through thermodynamic equilibrium and the K-value correlations you select, treating water as a miscible component.

Always assume water is immiscible

This selection specifies DESIGN II may treat water as a separate liquid phase, if present in sufficient quantity. It is used for systems containing oil and water.

Immiscibility assumption controlled by K-value method

The K-value correlation you select has a default setting for water miscibility or immiscibility. Water handling will be set automatically by DESIGN II.

Vapor saturation method/Combo Box

Select either Vapor Pressure or McKetta chart.

Vapor Pressure: Use the vapor pressure of water, corrected for pressure effects, to saturate the vapor phase of the mixture with water. The vapor phase will be saturated with water only if there is sufficient water to do so.

McKetta chart: Use the McKetta chart correlations to saturate the vapor phase of the mixture with water. The vapor phase will be saturated with water only if there is sufficient water to do so.

Liquid saturation method/Combo Box

Select either Kerosene chart or Light HC chart.

Kerosene chart: Use the API *Technical Data Book* Figure 9A1.4 kerosene chart to saturate the liquid oil phase of an oil and water mixture with water. The oil phase will be saturated only if there is sufficient water to form a water phase.

Light HC chart: Use the API *Technical Data Book* Figure 9A1.5-1 chart to saturate the liquid oil phase of a mixture with water. The oil phase will be saturated only if there is sufficient water to form a water phase.

Water Phase Flash: Turn Water Phase Flash Off

This options turns off the aqueous phase (liquid 2) flash and the component distribution will only have non-soluble water in the aqueous phase.

General Flash Method (used for all streams and unit modules if water is immiscible)

Select either Use the non-rigorous Three Phase (vapor – liquid 1 – liquid 2) Flash Method or Use the rigorous Three Phase (vapor – liquid 1- liquid 2) Flash Method (default)

Peng-Robinson

Use Peng-Robinson Options to set Peng-Robinson equation options, providing an acceptable simulation of K-values for mixtures of low molecular weight hydrocarbons with default interaction parameters.

1. Open the **Specify** menu and select **Basic Thermo**. The Thermodynamic and Transport Methods dialog displays with the Thermodynamics tab selected. Select the **Peng-Robinson Options** tab.

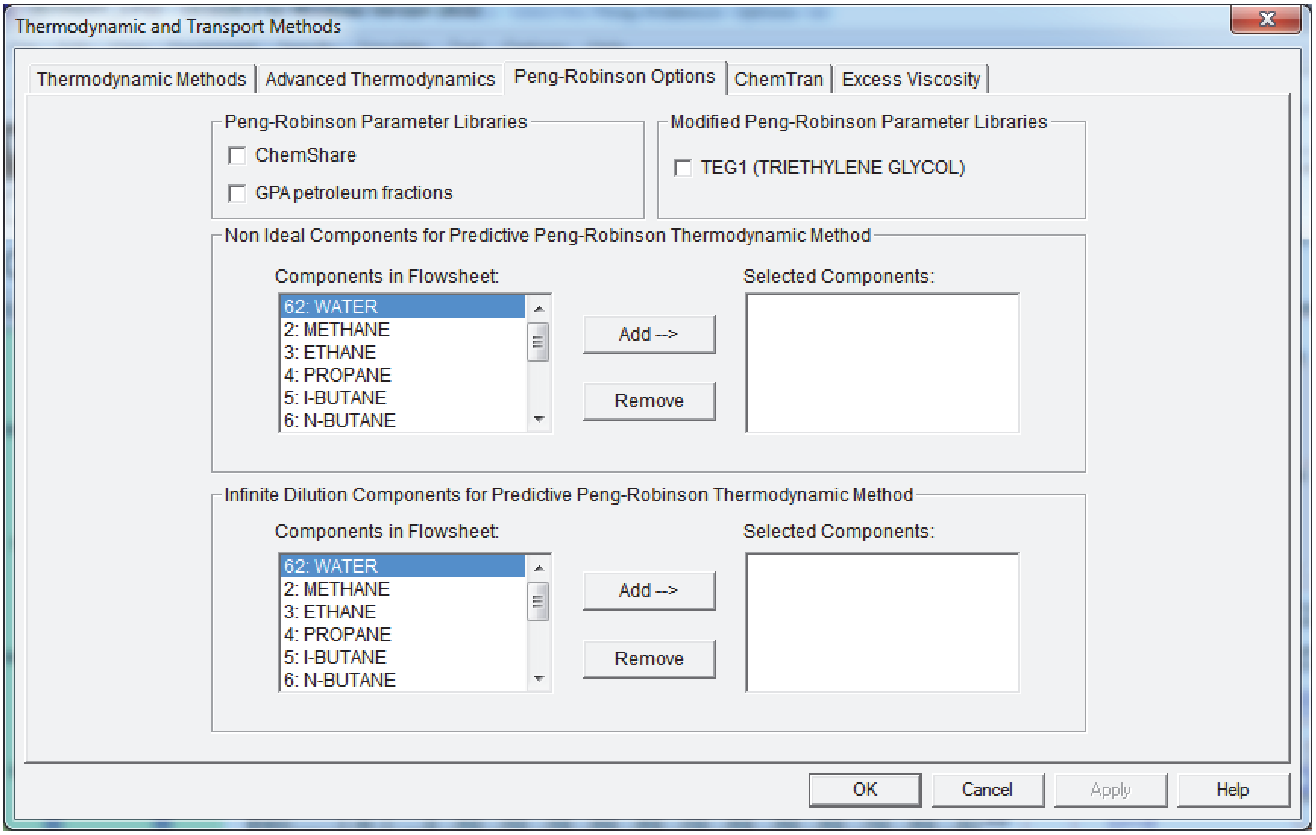


Figure 4: Peng-Robinson Options tab

Peng Robinson Parameter Libraries

When you use the Peng-Robinson option for K-values, enthalpies, and/or densities for mixtures, two libraries of binary interaction parameters are available. You can check **ChemShare** and/or **GPA petroleum fractions**. If you check both boxes, DESIGN II uses the ChemShare parameters and fills in with the GPA parameters for binaries involving petroleum fractions and the following components: nitrogen, carbon dioxide, hydrogen sulfide, methane, ethane, and propane.

Modified Peng Robinson Parameter Libraries

Select this checkbox to use TEG/Water/Hydrocarbons binary interaction parameters when the Modified Peng Robinson method is used

2. Select the desired option you want to use and complete the necessary information.

Non Ideal Components for Predictive Peng-Robinson Thermodynamic Method

Select the components that are to be treated as non-ideal. Click on a component from the list and click the Add button.

Infinite Dilution Components for Predictive Peng-Robinson Thermodynamic Method

Select the components that are to be treated as non-ideal. Click on a component from the list and click the Add button.

Ionic Components

DESIGN II has implemented the Edwards et al. method for weak aqueous electrolytes. This method requires many thermophysical properties for the components and ions present in the solution.Use Ionic Components/Reactions to select ionic components and reactions to use.

1. Open the **Specify** menu and select **Components**. The Components dialog displays with the Components tab selected. Select the **Ionic Components/Reactions** tab.

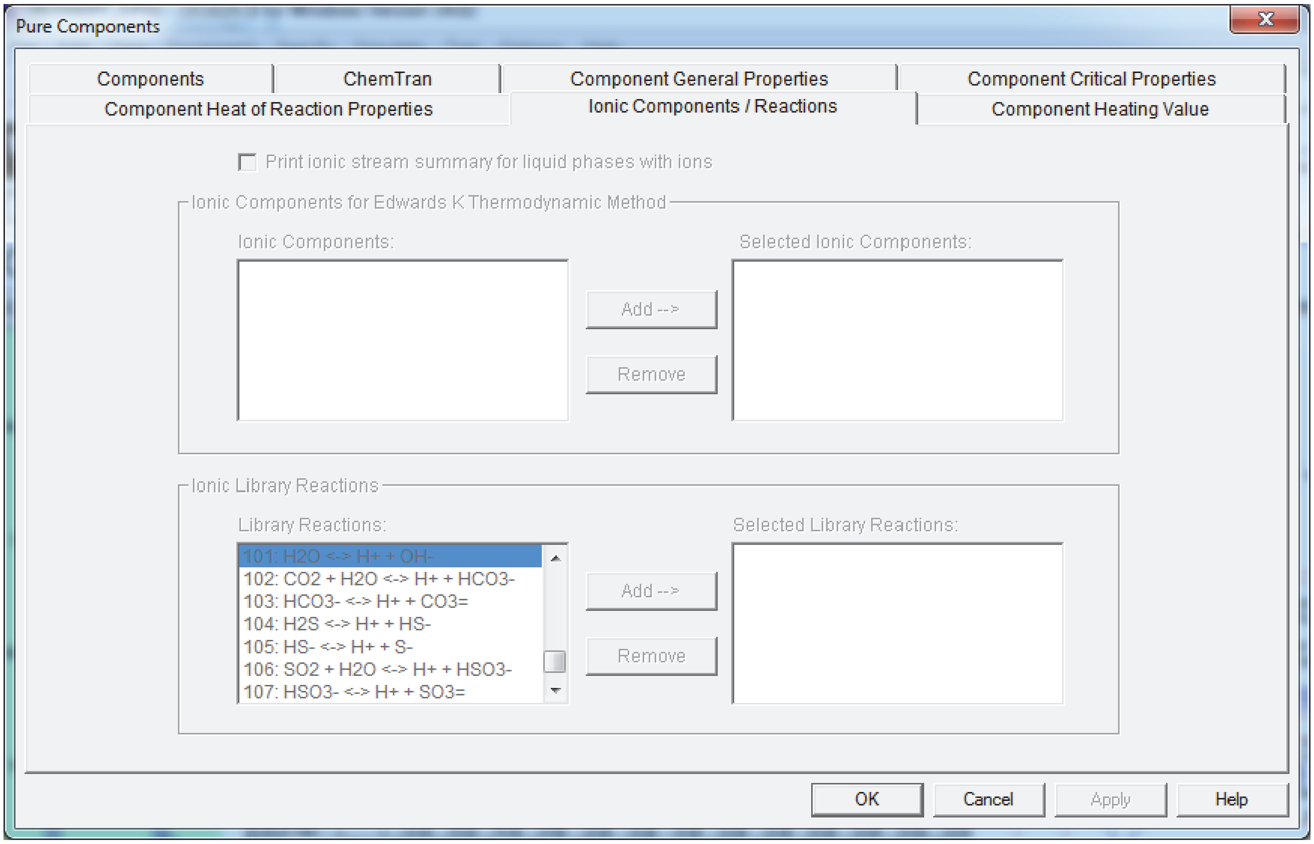


Figure 5: Ionic components/Calculations tab

Print ionic stream summary for liquid phases with ions

Select this check box to request a report detailing the ionic composition of each stream in the flowsheet. This information will print in the Stream Summary section of the output. The stream summary report and ionic stream reports will be merged (4 streams with regular data, 4 streams with ionic report, etc.)

Ionic Components for Edwards K Thermodynamic Method

Click on the component; then click the Add button. It is displayed in the Selected Ionic Components list. To remove a component from this list, click on it then click the Remove button.

Ionic Library Reactions

Click on the reaction; then click the Add button. It is displayed in the Selected Library Reactions list. To remove a reaction from this list, click on it then click the Remove button.

Assay Data / Crude Cuts

Use Assay Data to describe crude feeds. DESIGN II accepts crude feed streams in one of three ways, distinguished by the type of information provided:

1. Bulk feed analysis via ASTM or TBP laboratory distillation curves
2. Selection from the library of world crudes
3. Component-by-Component specification

Each feed (or recycle) stream on the flowsheet may be described using a different method. Describing a crude feed using ASTM/TBP laboratory distillation curves or the library of world crudes is covered in the ***Streams*** chapter.

If you select the component-by-component basis, you ***must*** enter mean average boiling point and gravity values for each pseudocomponent. If you select the distillation curve feed description basis, you may enter mean average boiling point values, if desired.

Additionally, you may enter component-by-component specification of average molecular weights for the pseudocomponents.

Cuts

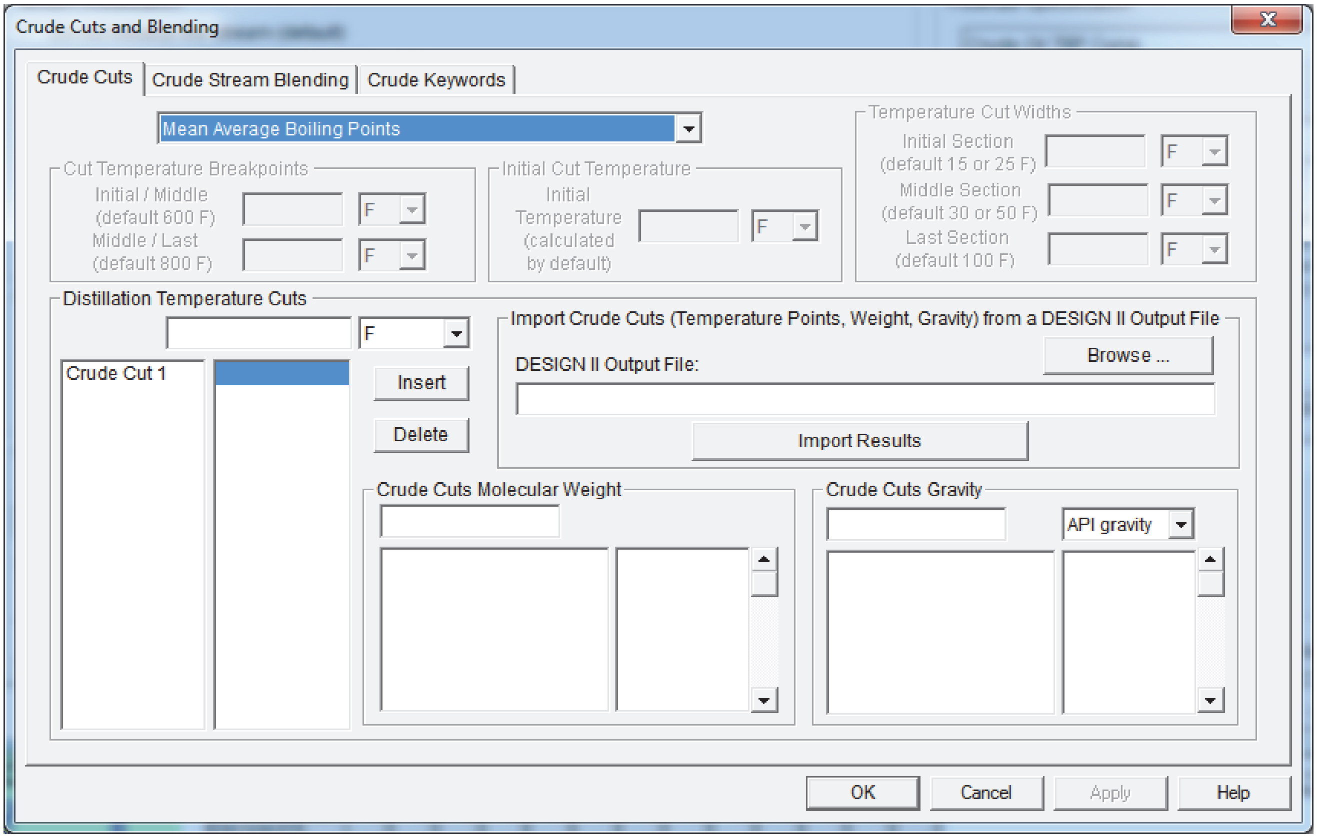


Figure 6: Crude Cuts tab (from refinery.psd)

1. Double click on the desired stream. The Stream dialog appears with the Stream Specifications tab selected. Select the Crude Cuts and Blending button. The dialog will open with the Crude Cuts tab selected.
2. Click on the drop down list and select Mean Average Boiling Points; then enter values for the cuts.

   If you characterize a stream by specifying component-by-component data; then you must specify individual boiling points for the cuts by selecting Mean Average Boiling Points.

   The number of cuts is very important for an adequate feed description; it is easier to calculate clear product distillation curves if you use many or narrow cuts.

   If you characterize all of the feed streams by distillation curves, DESIGN II first removes the specified volume percents for any light end components; then automatically divides the rest of the distillation curves into pseudocomponents, assigning each a boiling point, flowrate, average gravity, and molecular weight.

   If the TBP endpoint (100% volume) of the distillation curve is less than 1000 F, DESIGN II will create pseudocomponents with average boiling points 15o apart up to 600 F, and average boiling points 30 F apart after 600 F. For cuts above 800 F, the average boiling points are 100 apart.

   If the TBP endpoint is greater than 1000 F, average boiling points 25 F apart are assigned for pseudocomponents up to 600 F, average boiling points 50 F apart for temperatures between 600 F and 800 F, and 100 F apart for temperatures above 800 F on the TBP curve.

   In many cases you will find it necessary to adjust this pseudocomponent slate. For example, if too many pseudocomponents are created (making it computationally expensive) or you want more cuts in the middle portion of the feed and fewer in the heavy ends, specify your own pseudocomponent slate. Regardless of the endpoints of the TBP curve, the highest value you should enter for a pseudocomponent boiling point is 1800 F.

   The combo box on the dialog provides two ways of specifying the pseudocomponent slate: either by selecting Mean Average Boiling Points or Distillation Curve Breakpoints; the default option, , allows DESIGN II to determine the average boiling points for the pseudocomponent slate for feeds described with distillation curves.

   When you select Mean Average Boiling Points, a Vector Column displays. Enter the mean average boiling points in the Vector Column in ascending order from the initial boiling point to the end point.

   When you select Distillation Curve Breakpoints, enter five parameters using the edit boxes and their adjacent unit selection combos. After entering the parameters, cuts of Initial Section Cut Width are created up to the Initial/Middle Temperature Breakpoint value; cuts of Middle Section Cut Width are created between the Initial/Middle Temperature

Breakpoint and Middle/Last Temperature Breakpoint values; and cuts of Last Section Cut Width are created above the Middle/Last Temperature Breakpoint value.

Select the No Cut Points choice to allow DESIGN II to determine the default pseudocomponent slate for feeds described with distillation curves.

Refer to the on-line help file for details.

3. Enter the pseudocomponent gravity values. A gravity specification is required to characterize pseudocomponents; do this by specifying gravity values for each pseudocomponent.

   If all streams have the same gravity for the corresponding cut, enter the Pseudocomponent Gravities on this dialog. If you are combining streams which have different gravities for the corresponding cuts, enter the Pseudocomponent Gravities individually for each Stream using the Stream-Basics dialog.

   A Linked List Box is provided for entering the pseudocomponent gravities. The left column of the table shows the average boiling points representing the pseudocomponents. Enter the corresponding average gravities in the right column of the table for each pseudocomponent.

4. Enter the desired molecular weights corresponding to the cuts. DESIGN II automatically assigns molecular weights to each pseudocomponent using the correlation technique of Hariu. This correlation is reasonable for "light" crudes and most lean oils.

   Override this by entering the molecular weight for each pseudocomponent. If all streams have the same molecular weight for the corresponding cuts; then enter the Pseudocomponent Molecular Weights on this dialog.

   If you are combining streams which have different molecular weights for the corresponding cuts; then enter the Pseudocomponent Molecular Weights individually for each Stream using the Stream-Basics dialog.

   Linked list boxes are provided for entering the pseudocomponent molecular weights. The left column of the table shows the average boiling points representing the pseudocomponents. Enter the corresponding average molecular weights in the right column of the table for each pseudocomponent.

5. Click **OK** when done. The Crude Cuts and Blending dialog closes.

Keyword Input

1. Double click on the desired stream. The Stream dialog appears with the Stream Specifications tab selected. Select the Crude Cuts and Blending button. When the Crude Cuts and Blending dialog opens, select Crude Keywords tab.

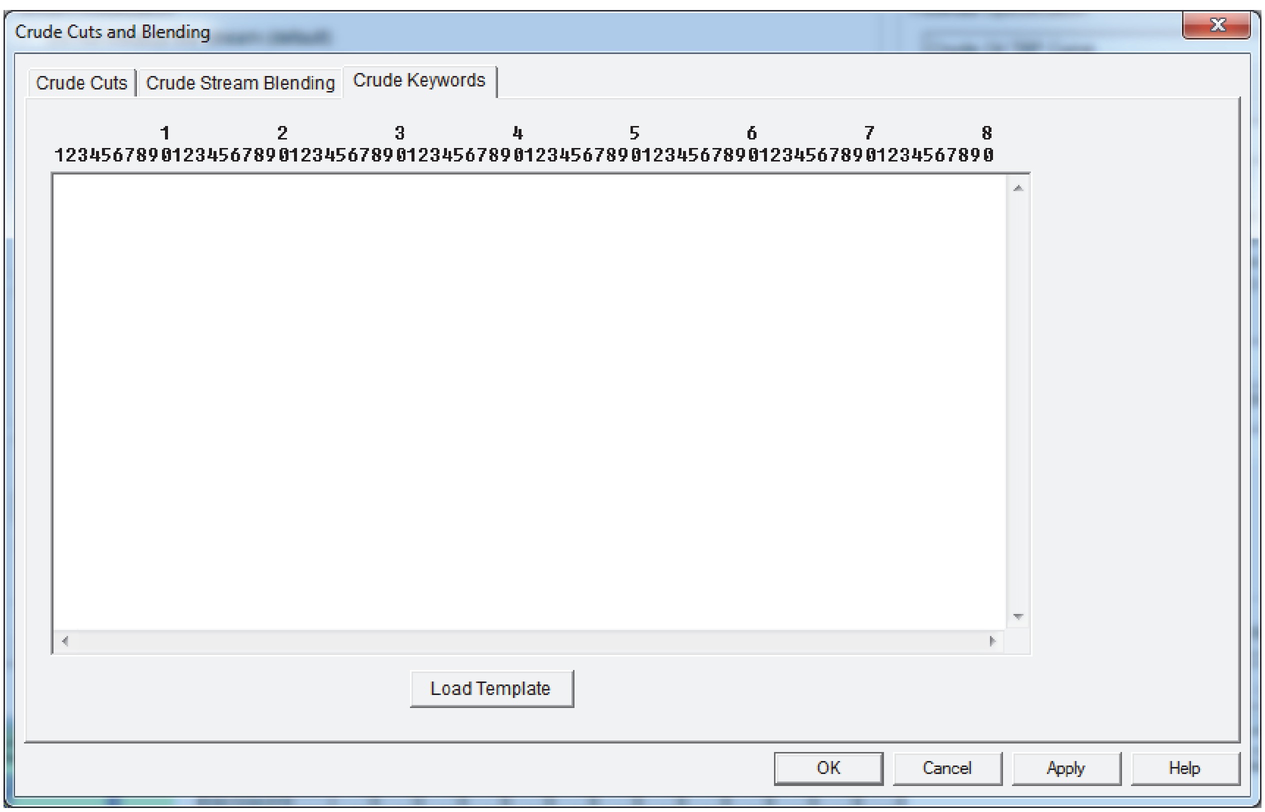


Figure 7: Crude Keywords tab

2. Enter the desired keyword input, or click the **Load Template** button to load a pre-defined template of keywords.

This dialog is similar to the one used for setting Optional General specs: Keyword Input. Refer to the ***Setting Optional General specs: Keyword Input***, ***Creating Keyword Commands***, and ***Editing Keyword Commands*** sections later in this chapter for details.

3. Click **OK** when done. The Crude Cuts and Blending dialog closes.

Crude Stream Blending

You can designate how crude streams will be blended with feed streams. All feed streams and specified streams will be automatically blended in the first crude blend unless otherwise specified. For each crude blend, you can have up to 200 crude stream blends and 200 feed streams.

1. Double click on the desired stream. The Stream dialog appears with the Stream Specifications tab selected. Select the Crude Cuts and Blending button. When the Crude Cuts and Blending dialog opens, select Crude Stream Blending tab.

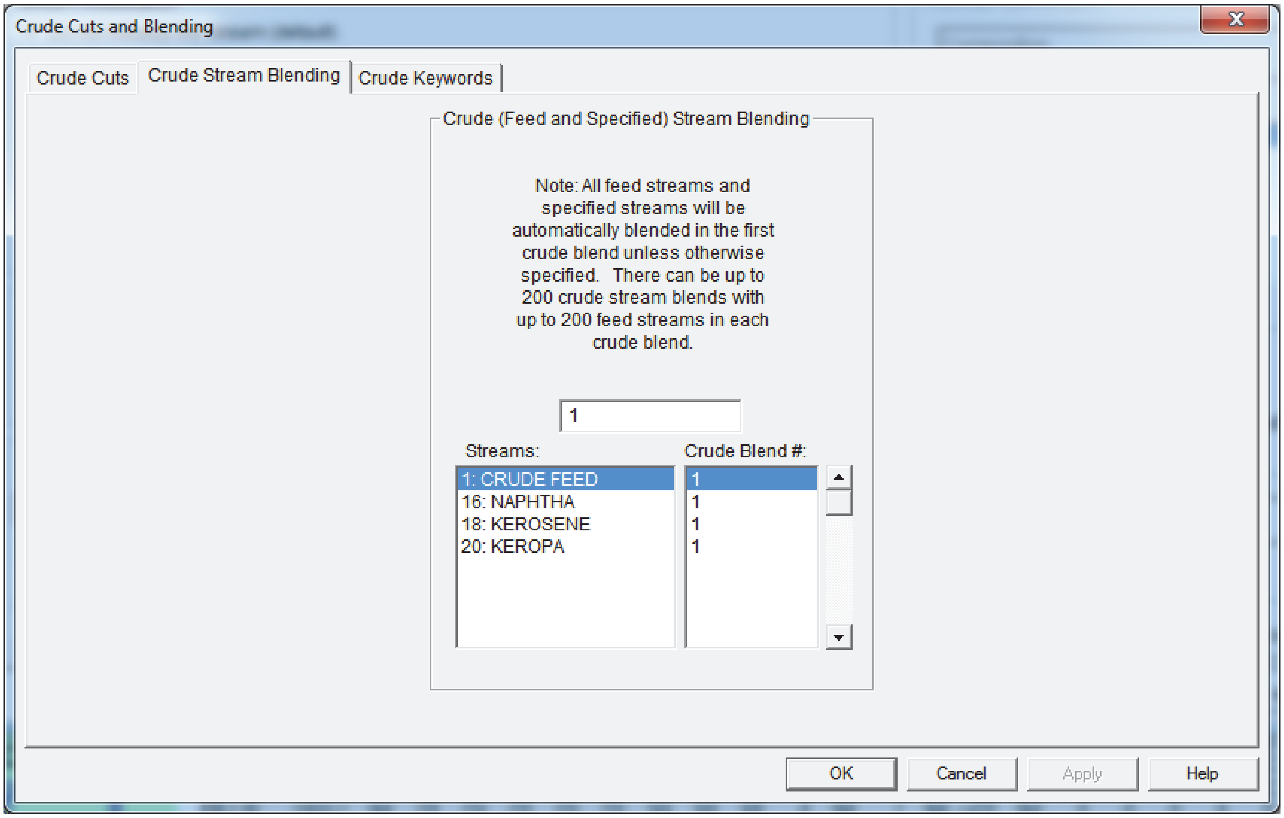


Figure 8: Crude Stream Blending tab tab (from httrain.psd)

2. Click on a stream name from the the list and enter a blend number (between 1 and 200).

3. Click **OK** when done. The Crude Cuts and Blending dialog closes.

ChemTran

Accurate physical property and thermodynamic data are essential for a process simulator to provide accurate process design, equipment sizing, energy balance, and mass balance of calculations. ChemTran provides the ability to define, regress, and predict all thermophysical properties necessary for accurate process simulation calculations.

ChemTran is designed to integrate experimental data and physical property prediction methods with DESIGN II data correlations to improve their accuracy and increase their applicability. DESIGN II for Windows allows you to create ChemTran input files using one of three methods to generate input: 1) by dialog, 2) by keywords, or 3) an ASCII editor (blank text window).

In many applications, it is not necessary to use ChemTran to accurately model a process with DESIGN II. However, ChemTran may be required to augment DESIGN II's physical property database and thermodynamic correlations when modeling processes involving non ideal chemical systems, proprietary chemical components, or extreme process conditions.

To use the dialog or keyword methods to create a ChemTran input file:

1. Open the **Specify** menu and select **Components**. The Components dialog appears with the Components tab selected. Click on the ChemTran tab.

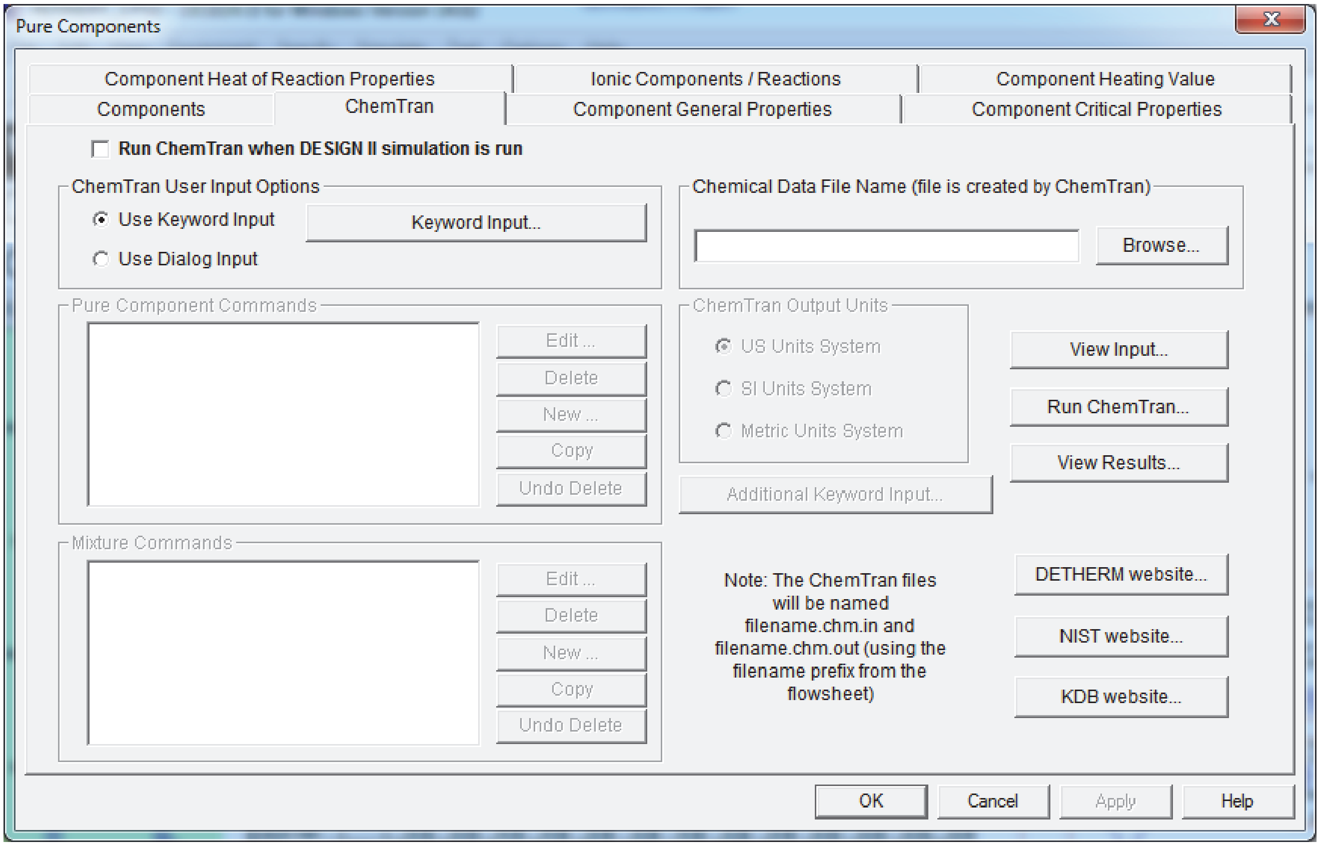


Figure 9: ChemTran tab

2. Click the checkbox to Run ChemTran when DESIGN II simulation is run; then select the input method. You can also import an existing ChemTran data file to use (generated by ChemTran with an extension of .dat). You can view input, run ChemTran, and view the results. You can also launch related websites if you have an Internet connection.

3. If you selected Keyword Input, click the Keyword Input button and complete the ChemTran Keyword Input dialog; refer to the DESIGN II for Windows help system for details.

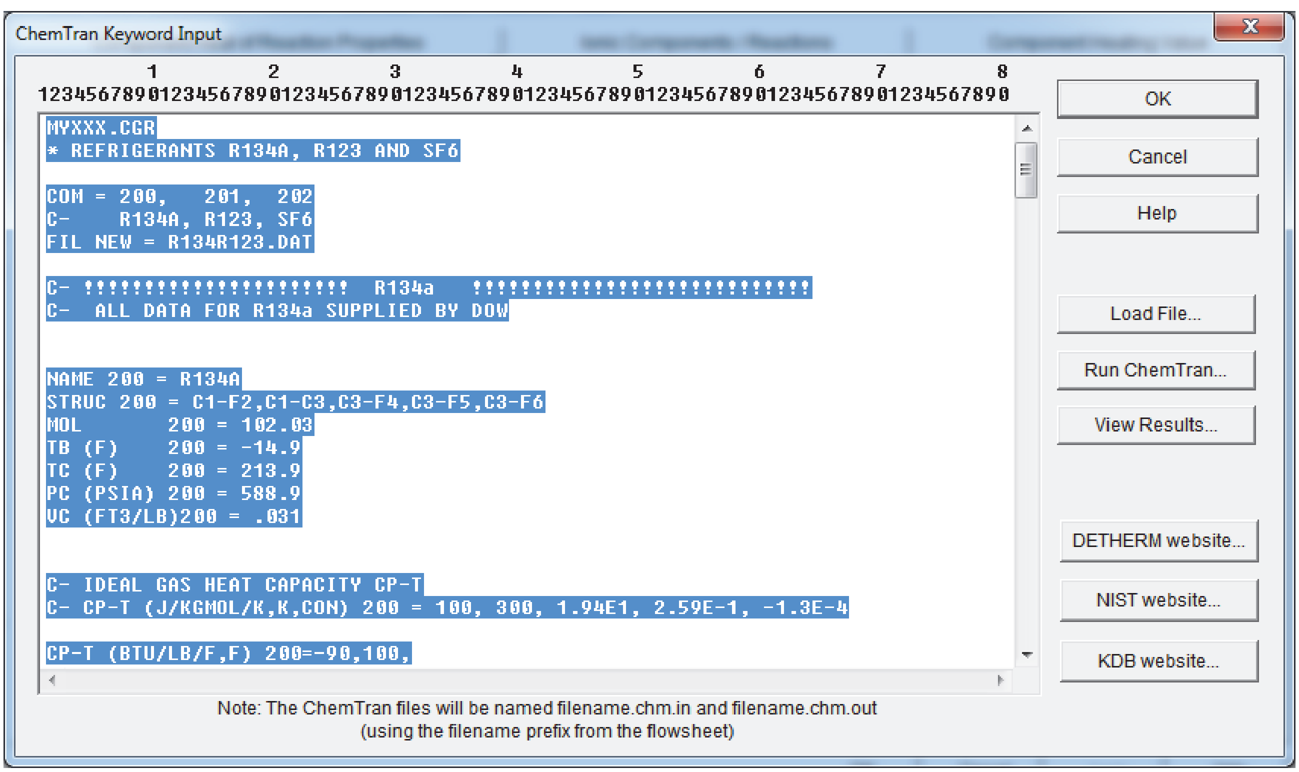


Figure 10: ChemTran Keyword Input dialog (using ChemTran1.psd)

4. If you selected Use Dialog Input, complete the Pure Component Commands dialog and Mixture Commands dialog. (You can also add additional keyword input as necessary.) Refer to the steps below for details.
5. For the dialog input method, you can use the Pure Component Commands button to display a dialog for creating and editing commands, copying existing ones, or deleting them. When you click the New or Edit button, the Pure Component Commands dialog appears.

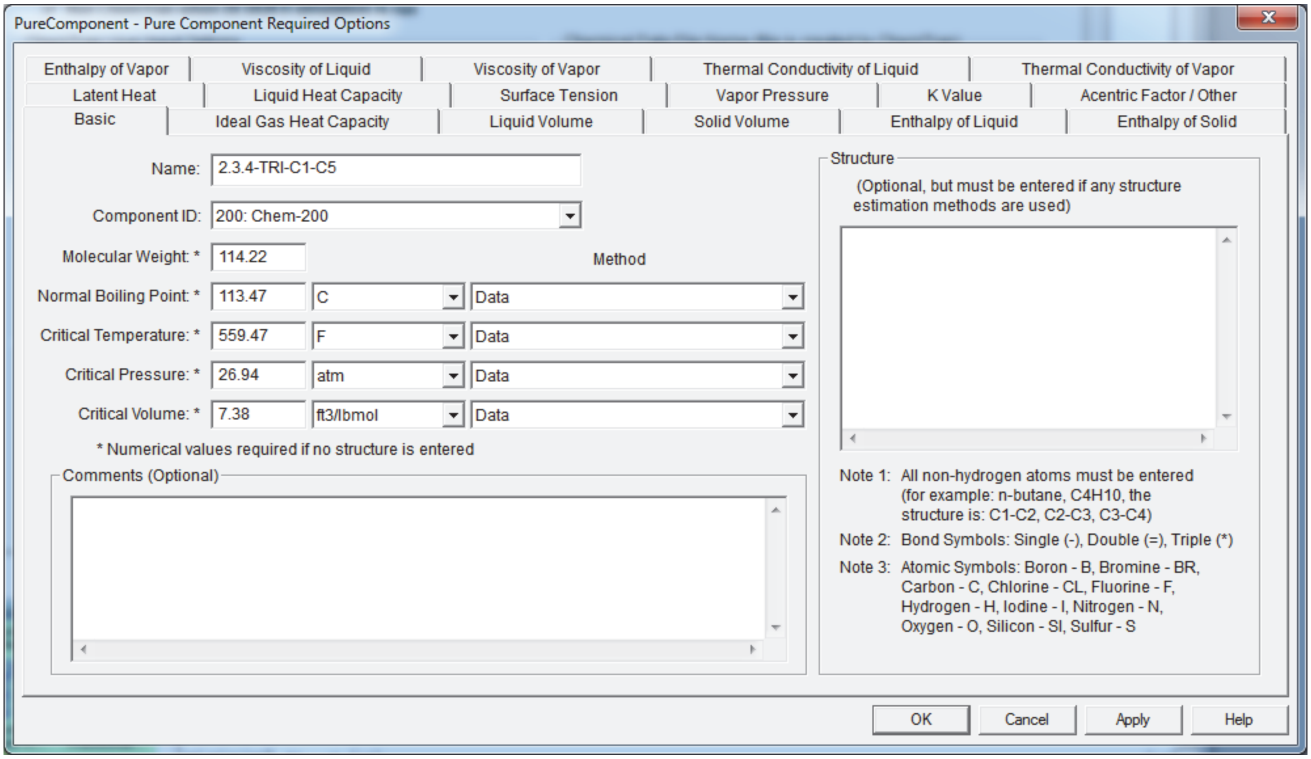


Figure 11: Pure Component Properties dialog (using ChemTran_PureComp & Mixture Dialog.psd)

6. Complete the following tabs as necessary:

Basic

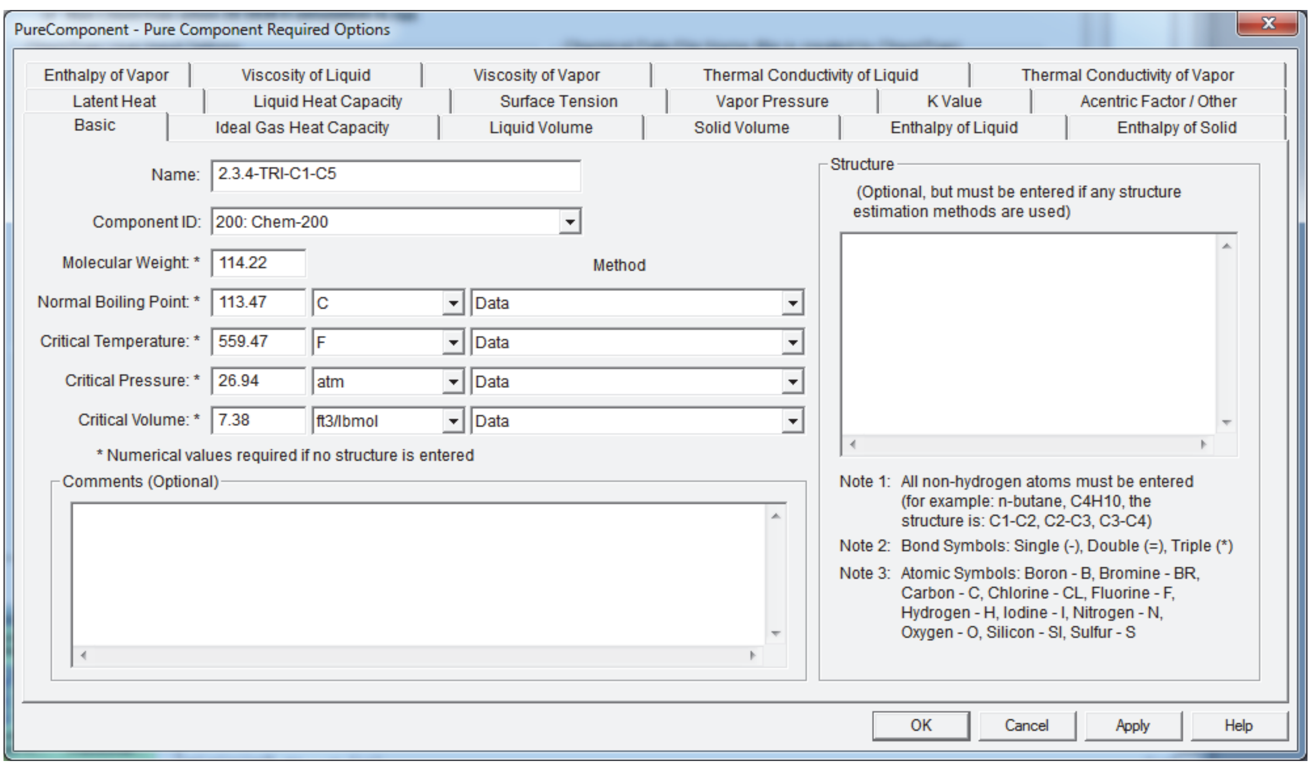


Figure 12: Basic tab (using ChemTran_PureComp & Mixture Dialog.psd)

ChemTran provides capability to enter, regress, or estimate pure component physical property data. These properties are stored in a user defined database, used to supplement the DESIGN II standard pure component physical property database.

Name: View the name of the component.

Component ID: Select the component ID from the drop-down list.

Molecular Weight: Enter the molecular weight*.

Normal Boiling Point: If you are using the Data method (default)*, enter a value and select a unit. If you need to use any structure estimation method, open the drop-down list under Method and choose the desired method to use: Ogata, Stiel-Thodos, or Lyderson-Forman-Thodos.

Critical Temperature: If you are using the Data method (default)*, enter a value and select a unit. If you need to use any structure estimation method, open the drop-down list under Method and choose the desired method to use: Lyderson or Forman-Thodos.

Critical Pressure: If you are using the Data method (default)*, enter a value and select a unit. If you need to use any structure estimation method, open the drop-down list under Method and choose the desired method to use: Lyderson Molecular or Forman-Thodos.

Critical Volume: If you are using the Data method (default)*, enter a value and select a unit. If you need to use any structure estimation method, open the drop-down list under Method and choose the desired method to use: Lyderson or Calculate from TC and PC (Yen-Woods)

* You must enter a numerical value if you do not enter a structure. If you do enter a value, you can still optionally enter a structure.

Structure: You can enter a structure for the pure component here. Structures are optional if you entered numerical values for any of the following: Molecular Weight, Normal Boiling Point, Critical Temperature, Critical Pressure, or Critical Volume. However, they are required if you selected any structure estimation method. Please review the three notes on the dialog beneath the Structure field for requirements and symbol conventions.

Comments: Enter any additional notes about pure component properties, if desired.

Ideal Gas Heat Capacity

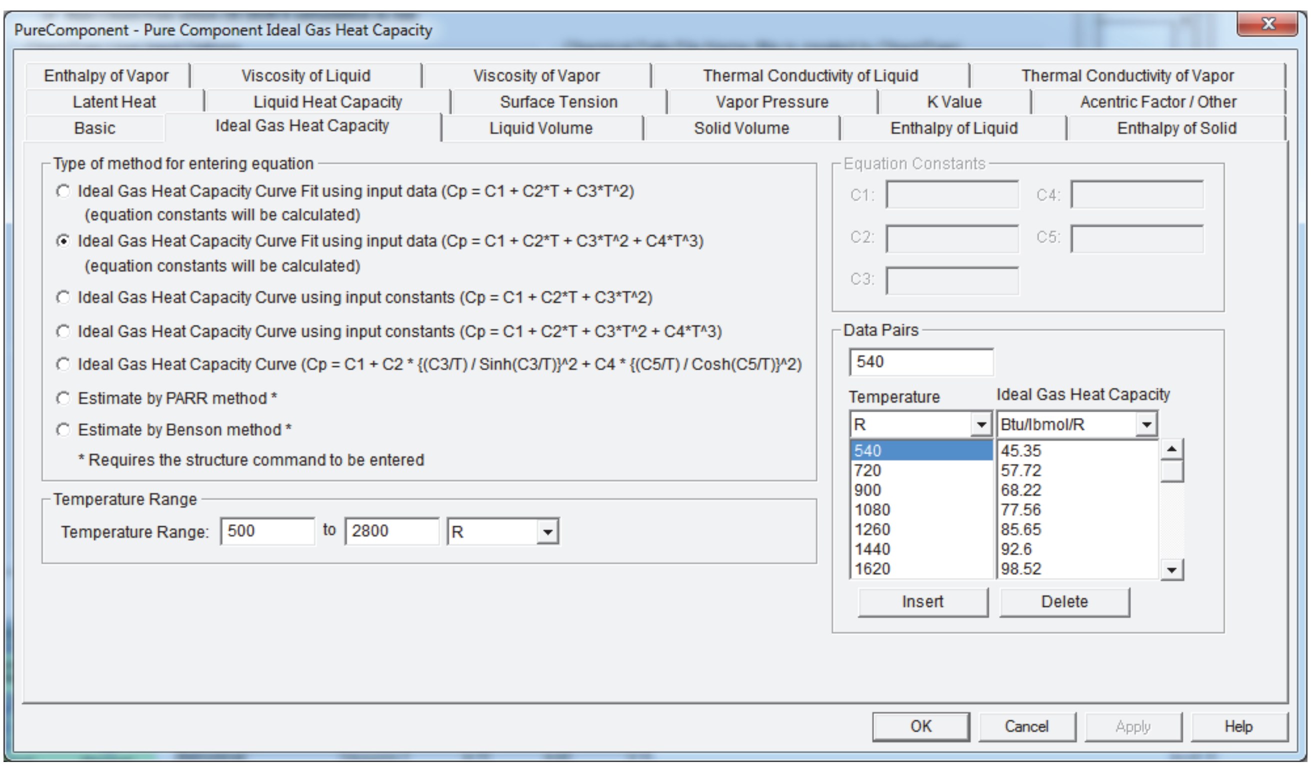


Figure 13: Ideal Gas Heat Capacity tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can select the type of method for estimating the ideal gas heat capacities for a molecule. There are two capacity curve fit equations using input data and three capacity curves using input constants. Two group contribution techniques generally referred to as the Benson method and Parr method are also available. The Benson method, as a general rule, will provide somewhat better results than the Parr method when it is applicable.

Type of method for entering equation: Select one of the following:

Ideal Gas Heat Capacity Curve Fit using input data (Cp = C1 + C2*T + C3*T^2): the equation constants will be calculated.

Ideal Gas Heat Capacity Curve Fit using input data (Cp = C1 + C2*T + C3*T^2 + C4*T^3): the equation constants will be calculated.

Ideal Gas Heat Capacity Curve using input constants (Cp = C1 + C2*T + C3*T^2)

Ideal Gas Heat Capacity Curve using input constants (Cp = C1 + C2*T + C3*T^2 + C4*T^3)

Ideal Gas Heat Capacity Curve (Cp = C1 + C2* {(C3/T)/Sinh(C3/T)}^2 + C4 * {C5/T)/Cosh(C5/T)}^2)

Estimate by PARR method*

Estimate by Benson Method*

* These options require you to enter the structure command on the Basic tab.

Equation Constants: If you selected any of the Ideal Heat Capacity Curve using input constant choices; then enter values in C1-C5 (depending on which option you selected)

Temperature Range: If you selected either Ideal Gas Heat Capacity Curve Fit using input data choice, enter the lowest temperature limit value; then the highest and select a unit.

Data Pairs: If you selected either Ideal Gas Heat Capacity Curve Fit using input data choice: Click on the list box below the Temperature column name (underneath the unit drop down list); a blue highlight appears in the box. You can now type the desired value. Click the Insert button to add the value. Repeat this for the Ideal Heat Capacity value. You can create up to 50 data pairs. You can also select the units to use for Temperature and Ideal Gas Heat Capacity.

Liquid Volume

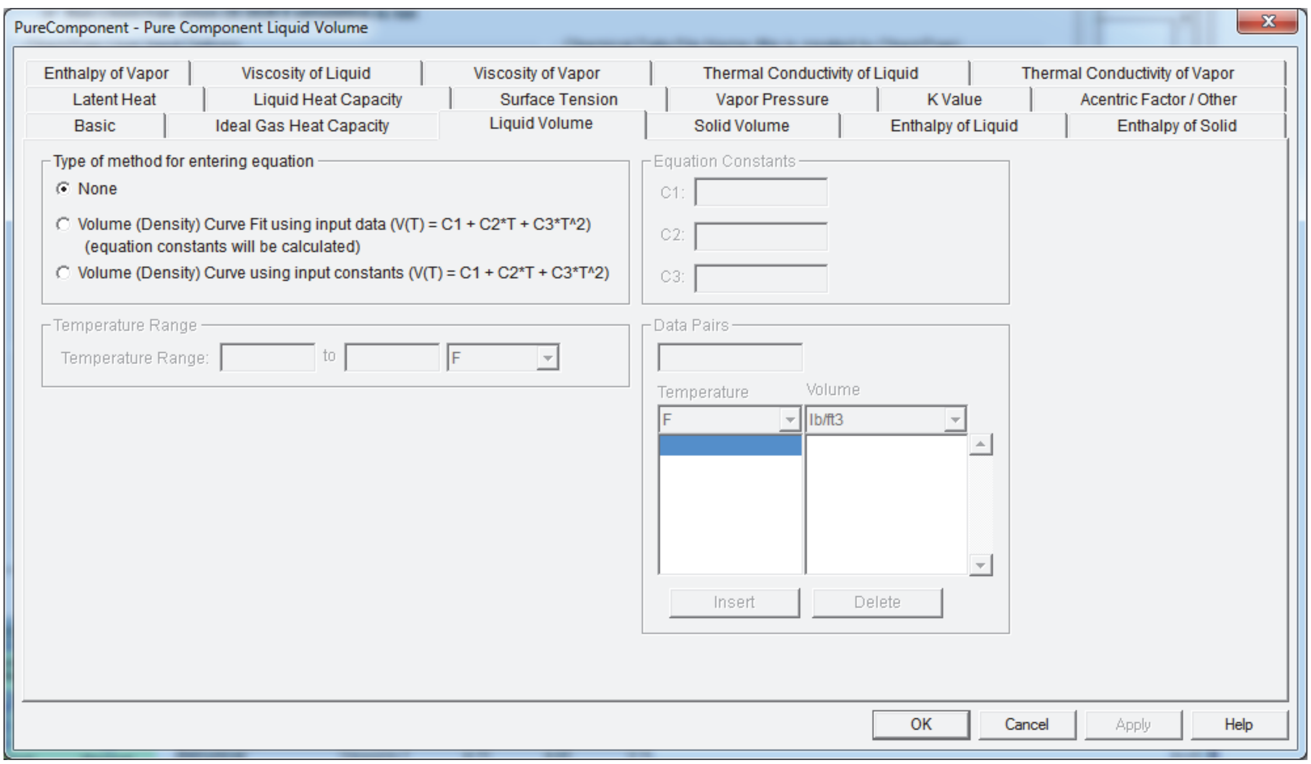


Figure 14: Liquid Volume tab (using ChemTran_PureComp & Mixture Dialog.psd)

Liquid densities (or volumes) are calculated as required by default options in ChemTran by a density correlation. Best results are obtained if liquid density or volume data is supplied for the component. You can select to enter volume-temperature data for curve fitting or the appropriate constants for an equation.

Type of method for entering equation: Select one of the following:

None

Volume (Density) Curve Fit using input data (V(T) = C1 + C2*T + C3*T^2)

Volume (Density) Curve using input constants (V(T) = C1 + C2*T + C3*T^2)

Equation Constants: If you selected the Volume (Density) Curve using input constants; then enter values in C1-C3.

Temperature Range: If you selected either Volume (Density) Curve option, enter the lowest temperature limit value; then the highest and select a unit.

Data Pairs: If you selected the Volume (Density) Curve Fit using input data choice, click on the list box below the Temperature column name (underneath the unit drop down list); a blue highlight appears in the box. You can now type the desired value. Click the Insert button to add the value. Repeat this for the Volume value. You can create up to 50 data pairs. You can also select the units to use for Temperature and Volume.

Solid Volume

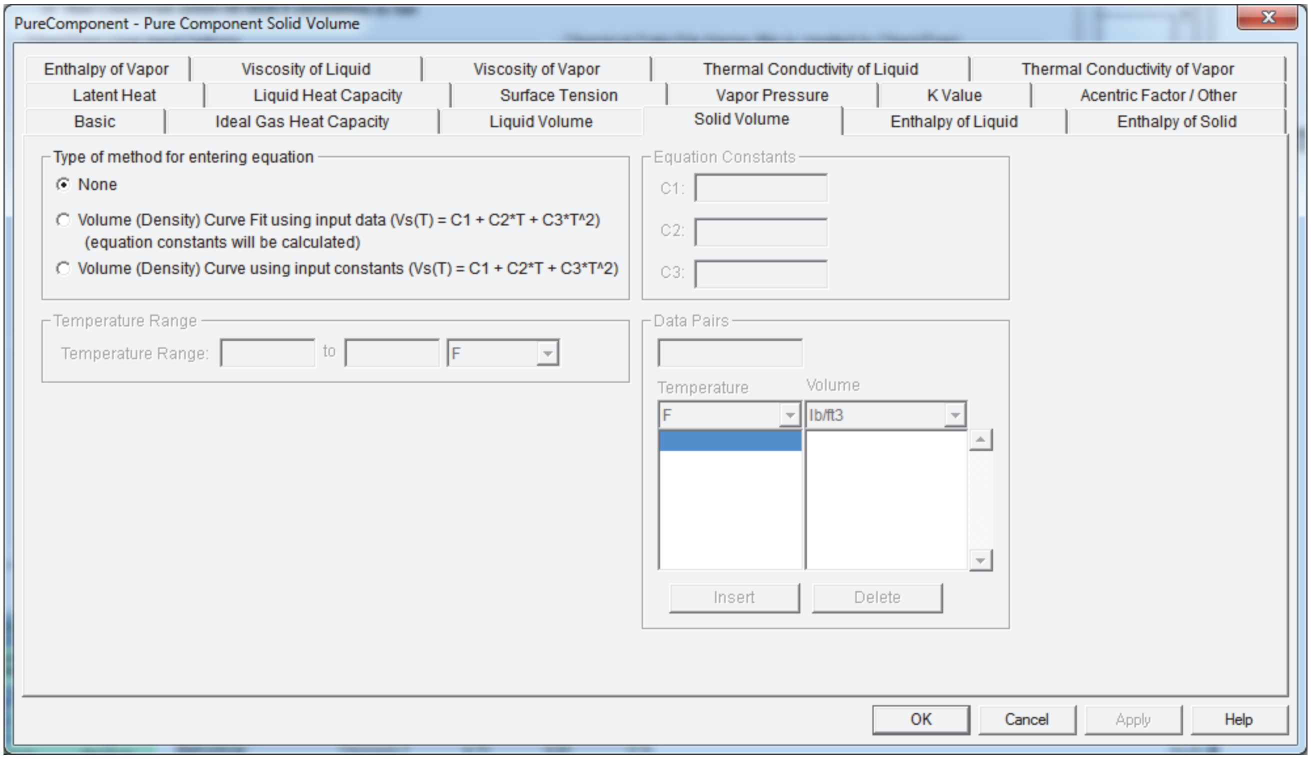


Figure 15: Solid Volume tab (using ChemTran_PureComp & Mixture Dialog.psd)

Solid densities (or volumes) are calculated as required by default options in ChemTran by a density correlation. Best results are obtained if solid density or volume data is supplied for the component. You can select the method for entering the solid volume equation. You can select to enter volume-temperature data for curve fitting or the appropriate constants for an equation using solid components.

Type of method for entering equation: Select one of the following:

None

Volume (Density) Curve Fit using input data (Vs(T) = C1 + C2*T + C3*T^2)

Volume (Density) Curve using input constants (Vs(T) = C1 + C2*T + C3*T^2)

Equation Constants: If you selected the Volume (Density) Curve using input constants; then enter values in C1-C3

Temperature Range: If you selected either Volume (Density) Curve option, enter the lowest temperature limit value; then the highest and select a unit.

Data Pairs: If you selected the Volume (Density) Curve Fit using input data choice: Click on the list box below the Temperature column name (underneath the unit drop down list); a blue highlight appears in the box. You can now type the desired value. Click the Insert button to add the value. Repeat this for the Volume value. You can create up to 50 data pairs. You can also select the units to use for Temperature and Volume.

Enthalpy of Liquid

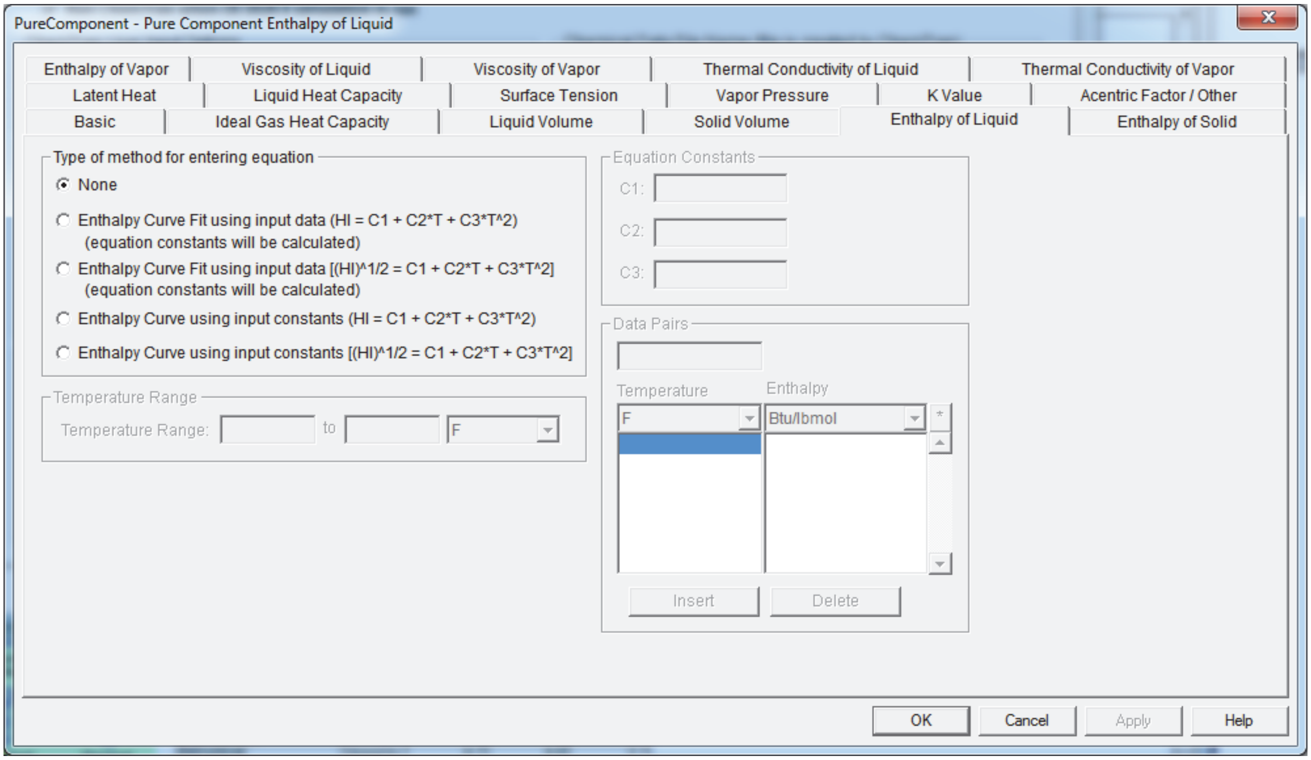


Figure 16: Enthalpy of Liquid tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can select the type of method to use for entering the Enthalpy of Liquid equation. You can enter liquid enthalpy versus temperature data for data reduction to one of two equations. Alternatively, you can enter the constants for the equation directly.

Type of method for entering equation: Select one of the following:

None

Enthalpy Curve Fit using input data (Hl = C1 + C2*T + C3*T^2)

Enthalpy Curve Fit using input data [(Hl)^1/2 = C1 + C2*T + C3*T^2]

Enthalpy Curve using input constants (Hl = C1 + C2*T + C3*T^2)

Enthalpy Curve using input constants [(Hl)^1/2 = C1 + C2*T + C3*T^2)

Equation Constants: If you selected either Enthalpy Curve using input constants choice; then enter values in C1-C3.

Temperature Range: If you selected any method other than None, enter the lowest temperature limit value; then the highest and select a unit.

Data Pairs: If you selected either Enthalpy Curve Fit using input data choice. Click on the list box below the Temperature column name (underneath the unit drop down list); a blue highlight appears in the box. You can now type the desired value. Click the Insert button to add the value. Repeat this for the Enthalpy value. You can create up to 50 data pairs. You can also select the units to use for Temperature and Enthalpy.

Enthalpy of Solid

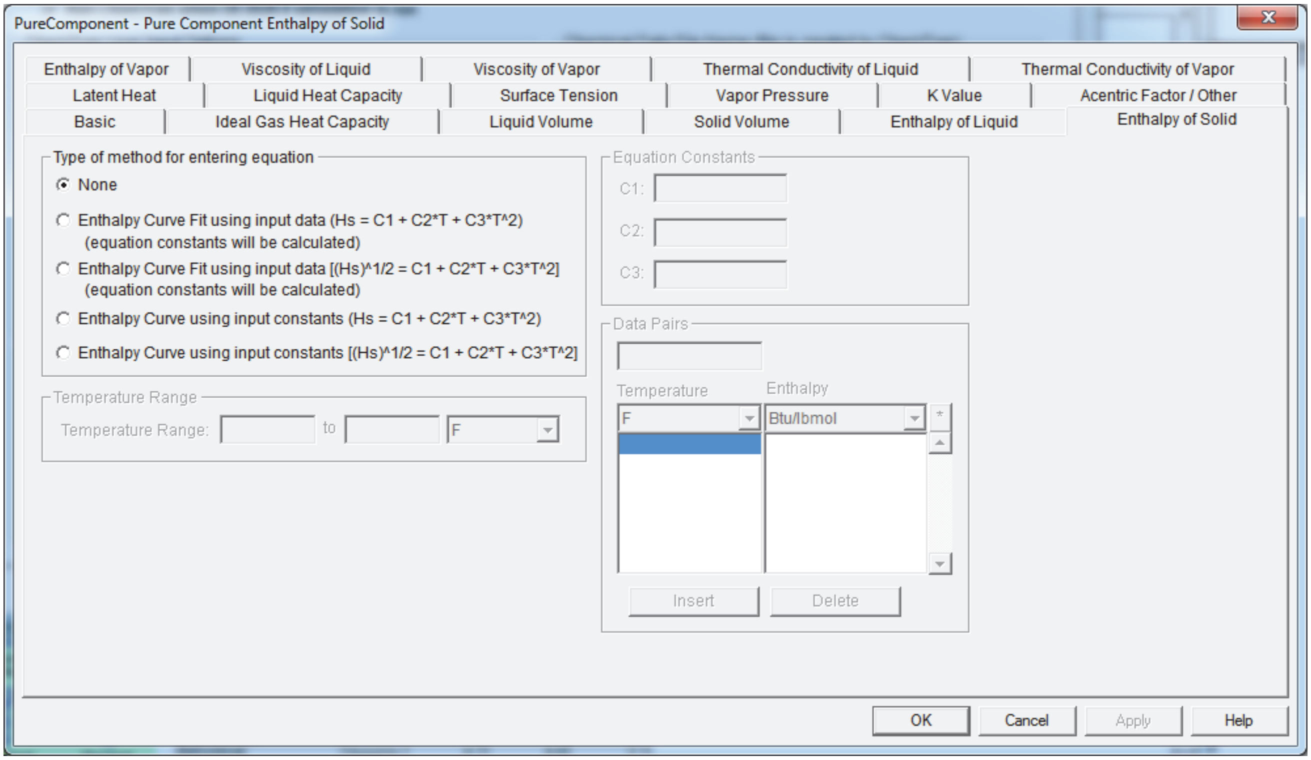


Figure 17: Enthalpy of Solid tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can select the type of method to use for entering the Enthalpy of Solid equation. You can enter solid enthalpy versus temperature data for data reduction to one of two equations. Alternatively, you can enter the constants for the equation directly.

Type of method for entering equation: Select one of the following:

None

Enthalpy Curve Fit using input data (Hs = C1 + C2*T + C3*T^2)

Enthalpy Curve Fit using input data [(Hs)^1/2 = C1 + C2*T + C3*T^2]

Enthalpy Curve using input constants (Hs = C1 + C2*T + C3*T^2)

Enthalpy Curve using input constants [(Hs)^1/2 = C1 + C2*T + C3*T^2)

Equation Constants: If you selected either Enthalpy Curve using input constants choice; then enter values in C1-C3.

Temperature Range: If you selected any method other than None, enter the lowest temperature limit value; then the highest and select a unit.

Data Pairs: If you selected either Enthalpy Curve Fit using input data choice, click on the list box below the Temperature column name (underneath the unit drop down list); a blue highlight appears in the box. You can now type the desired value. Click the Insert button to add the value. Repeat this for the Enthalpy value. You can create up to 50 data pairs. You can also select the units to use for Temperature and Enthalpy.

Latent Heat

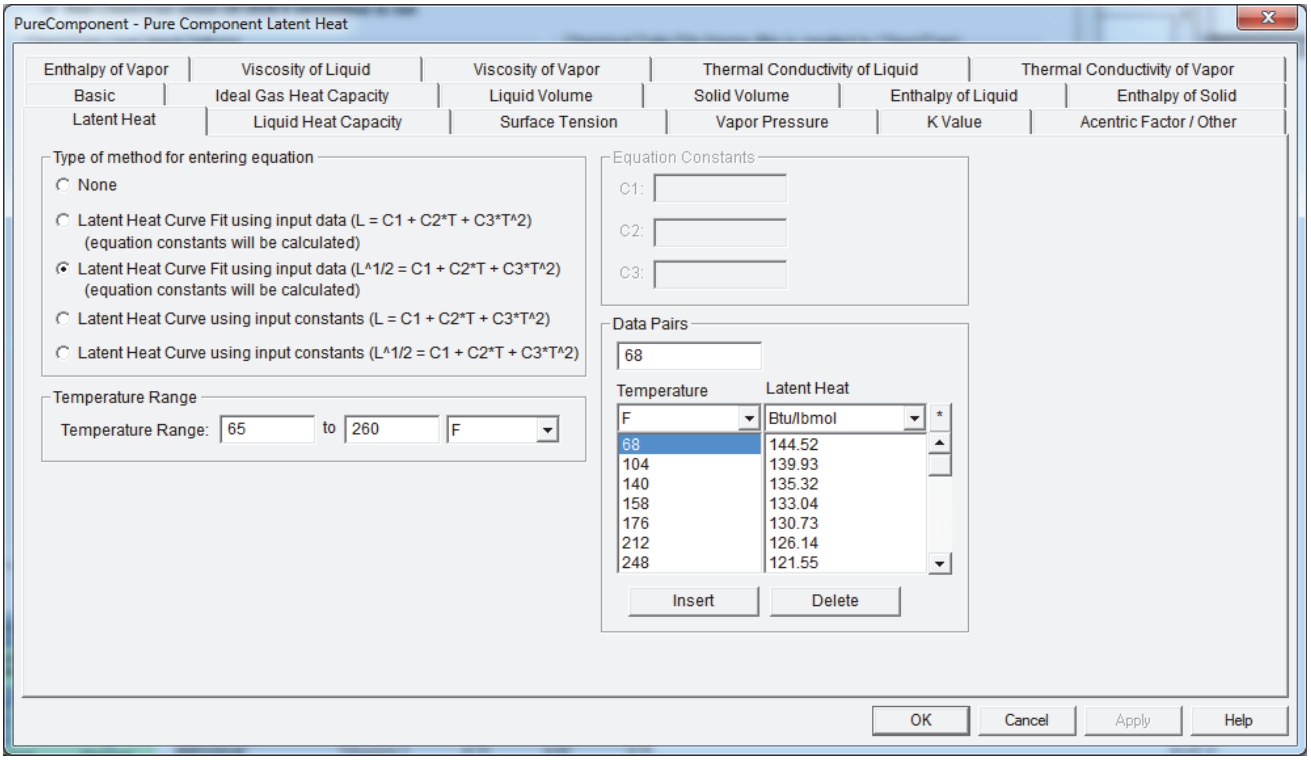


Figure 18: Latent Heat tab (using ChemTran_PureComp & Mixture Dialog.psd)

Sometimes latent heat data as a function of temperature is available in the absence of direct liquid and vapor enthalpy data. This method uses this data and an internal data system to calculate liquid enthalpy data from predicted vapor enthalpies. You can select the type of method to use for entering the Latent Heat equation. You can enter latent heat data (heat of vaporization) for curve fitting or constants for one of two equations. Latent heat data is used for calculating enthalpy of a liquid.

Type of method for entering equation: Select one of the following:

None

Latent Heat Curve Fit using input data (L = C1 + C2*T + C3*T^2)

Latent Heat Curve Fit using input data (L^1/2 = C1 + C2*T + C3*T^2)

Latent Heat Curve using input constants (L = C1 + C2*T + C3*T^2)

Latent Heat Curve using input constants (L^1/2 = C1 + C2*T + C3*T^2)

Equation Constants: If you selected either Latent Heat Curve using input constants choice; then enter values in C1-C3.

Temperature Range: If you selected any method other than None, enter the lowest temperature limit value; then the highest and select a unit.

Data Pairs: If you selected Latent Heat Curve Fit using input data, click on the list box below the Temperature column name (underneath the unit drop down list); a blue highlight appears in the box. You can now type the desired value. Click the Insert button to add the value. Repeat this for the Latent Heat value. You can create up to 50 data pairs. You can also select the units to use for Temperature and Latent Heat.

Liquid Heat Capacity

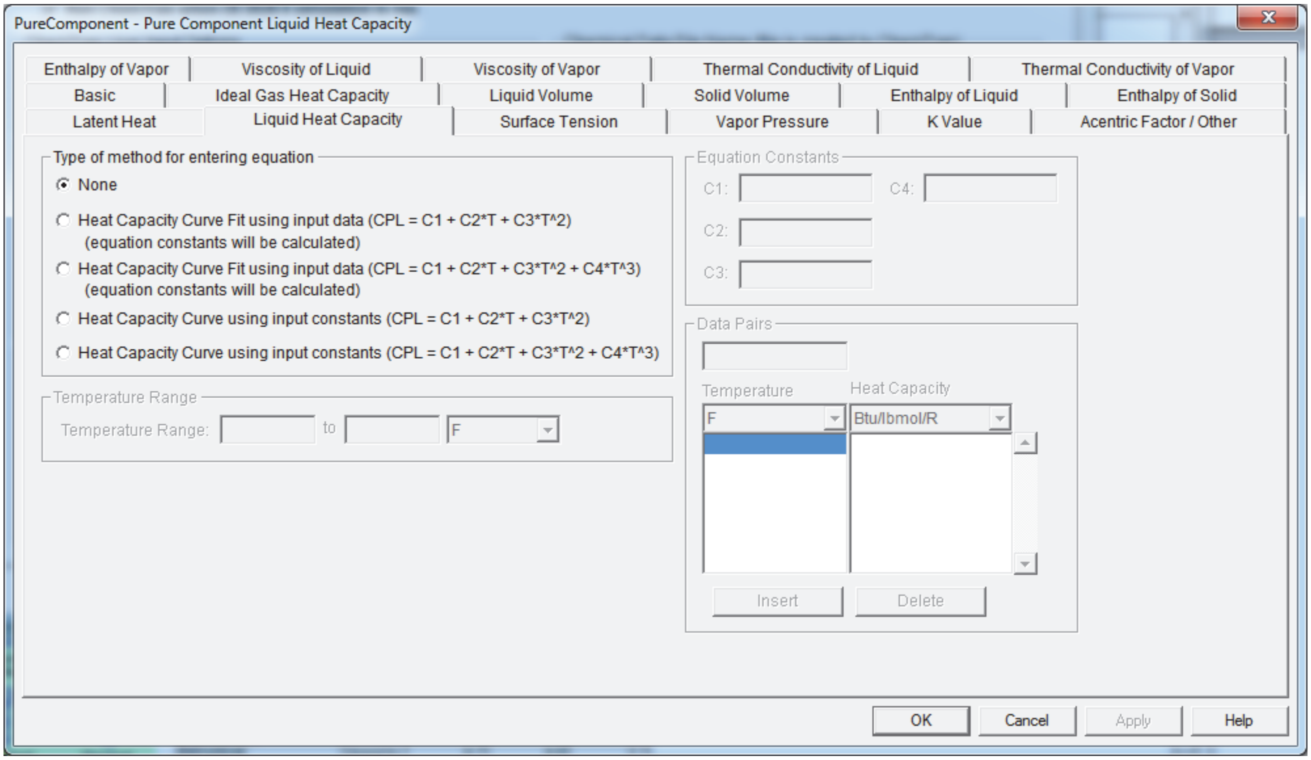


Figure 19: Liquid Heat Capacity tab (using ChemTran_PureComp & Mixture Dialog.psd)

Liquid heat capacity data is used for calculating enthalpy of a liquid. You can select the type of method to use for entering the Liquid Heat Capacity equation.

Type of method for entering equation: Select one of the following:

None

Heat Capacity Curve Fit using input data (CPL = C1 + C2*T + C3*T^2)

Heat Capacity Curve Fit using input data (CPL = C1 + C2*T + C3*T^2 + C4*T^3)

Heat Capacity Curve Fit using input constants (CPL = C1 + C2*T + C3*T^2)

Heat Capacity Curve Fit using input constants (CPL = C1 + C2*T + C3*T^2 + C4*T^3)

Equation Constants: If you selected either Heat Capacity Curve using input constants choice; then enter values in C1-C4.

Temperature Range: If you selected any method other than None, enter the lowest temperature limit value; then the highest and select a unit.

Data Pairs: If you selected either Heat Capacity Curve Fit using input data choice, click on the list box below the Temperature column name (underneath the unit drop down list); a blue highlight appears in the box. You can now type the desired value. Click the Insert button to add the value. Repeat this for the Heat Capacity value. You can create up to 50 data pairs. You can also select the units to use for Temperature and Heat Capacity.

Surface Tension

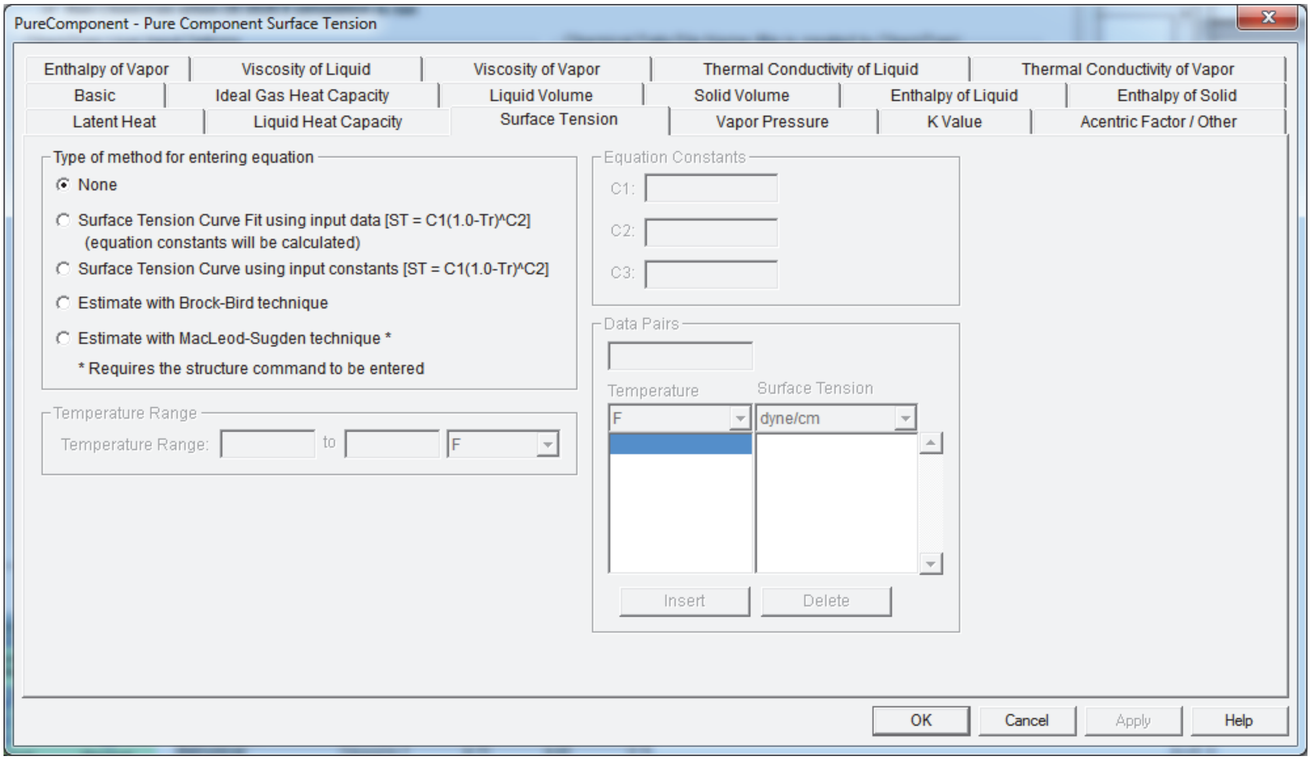


Figure 20: Surface Tension tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can select the type of method to use for entering the Surface Tension equation. You can enter surface tension data for a saturated liquid, or you can select estimation technique for the surface tension. The Brock-Bird technique for surface tension is based on a corresponding states analysis. It is comparable to the Macleod - Sugden technique for many chemicals but is not suited for chemicals exhibiting strong interactions. The Macleod-Sugden technique estimated surface tension for a component based on the parachor. The method is applicable to a wide variety of chemicals and is preferred where hydrogen bonding can occur. These include saturated and unsaturated hydrocarbons and hydrocarbons with the groups:

-COO-	-CHO
-COOH	O
-OH	N
$-NH_2$	S
-O-	P
$-NO_2$	F
$-NO_3$	Cl
$-CO(NH_2)$	B
=O	I

and many types of ring compounds.

Type of method for entering equation: Select one of the following:

None

Surface Tension Curve Fit using input data [ST = C1(1.0-Tr)^C2]

Surface Tension Curve Fit using input constants [ST = C1(1.0-Tr)^C2]

Estimate with Brock-Bird technique

Estimate with Macleod-Sugden technique: if you select this choice, you must enter a structure command on the Basic tab.

Equation Constants: If you selected the Surface Tension Curve using input constants; then enter values in C1-C3.

Temperature Range: If you selected either Surface Tension Curve choice, enter the lowest temperature limit value; then the highest and select a unit.

Data Pairs: If you selected the Surface Tension Curve Fit using input data choice, click on the list box below the Temperature column name (underneath the unit drop down list); a blue highlight appears in the box.

You can now type the desired value. Click the Insert button to add the value. Repeat this for the Surface Tension value. You can create up to 50 data pairs. You can also select the units to use for Temperature and Surface Tension.

Vapor Pressure

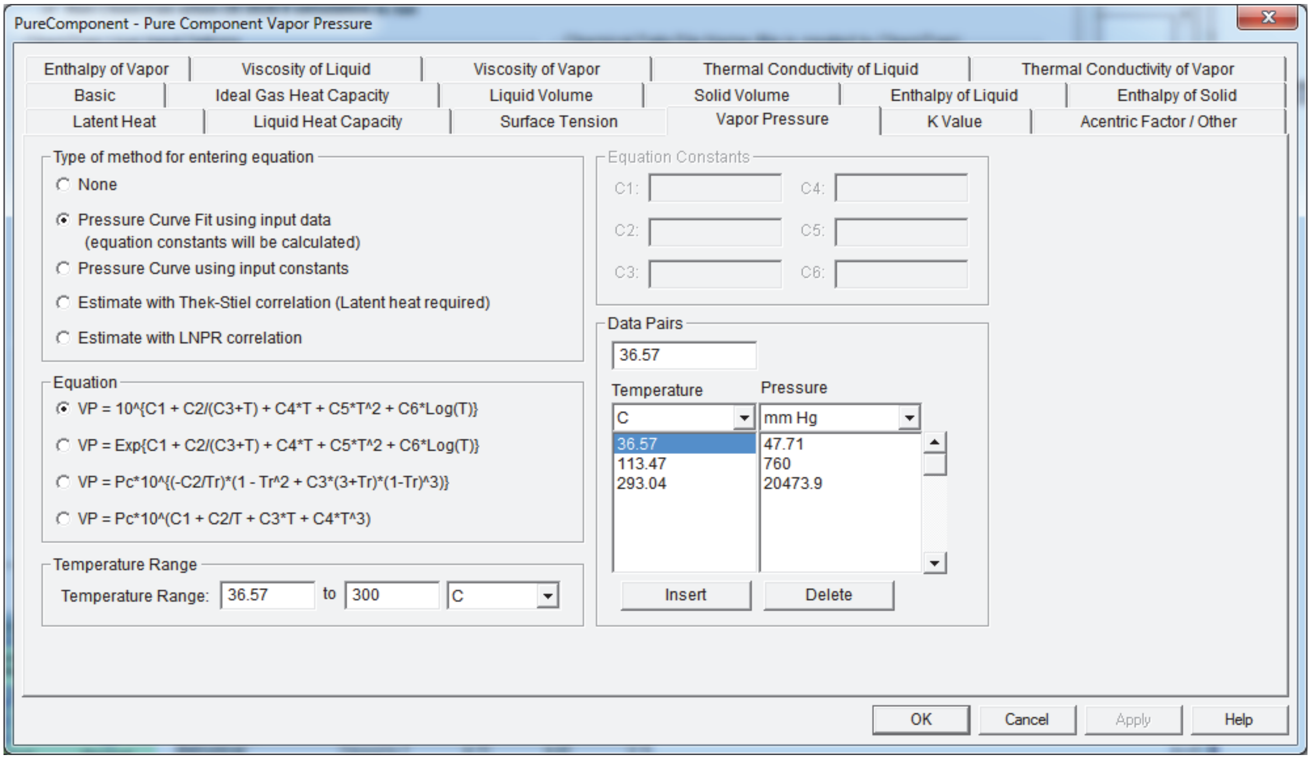


Figure 21: Vapor Pressure tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can select the type of method to use for entering the Vapor Pressure equation. You can enter vapor pressure versus temperature data for curve fitting, or constants for one of four vapor pressure equations. Vapor pressure data also can be estimated for a pure component in ChemTran by one of two methods: Thek-Stiel or LNPR. Both methods are based on correlations of data for a wide variety of compounds. Also, both methods require the user to enter other property data; you can select to utilize literature data or the estimation techniques furnished as options in ChemTran. These methods are used by ChemTran to calculate vapor pressures from the triple point to the critical temperature. A special default option is used if vapor pressures are required for supercritical conditions. A straight line, joining the boiling point and critical point, is used to calculate vapor pressures above the critical.

Type of method for entering equation: Select one of the following:

None

Pressure Curve Fit using input data

Pressure Curve using input constants

Estimate with Thek-Steil correlation (Latent heat required; refer to the Latent Heat tab for details)

Estimate with LNPR correlation

Equation Constants: If you selected Pressure Curve using input constants; then enter values in C1-C6.

Equation: If you selected either Pressure Curve choice, Select the equation to use:

VP = 10^{C1 + C2/(C3+T) + C4*T + C5*T^2 + C6*Log(T)}

VP = Exp{C1 + C2/(C3+T) + C4*T + C5*T^2 + C6*Log(T)}

VP = Pc*10^{(-C2/Tr)*(1 - Tr^2 + C3*(3+Tr)*(1-Tr)^3)}

VP = Pc*10^(C1 + C2/T + C3*T + C4*T^3)

Temperature Range: If you selected either Pressure Curve choice, enter the lowest temperature limit value; then the highest and select a unit.

Data Pairs: If you selected the Pressure Curve Fit using input data choice, click on the list box below the Temperature column name (underneath the unit drop down list); a blue highlight appears in the box. You can now type the desired value. Click the Insert button to add the value. Repeat this for the Pressure value. You can create up to 50 data pairs. You can also select the units to use for Temperature and Pressure.

K Value

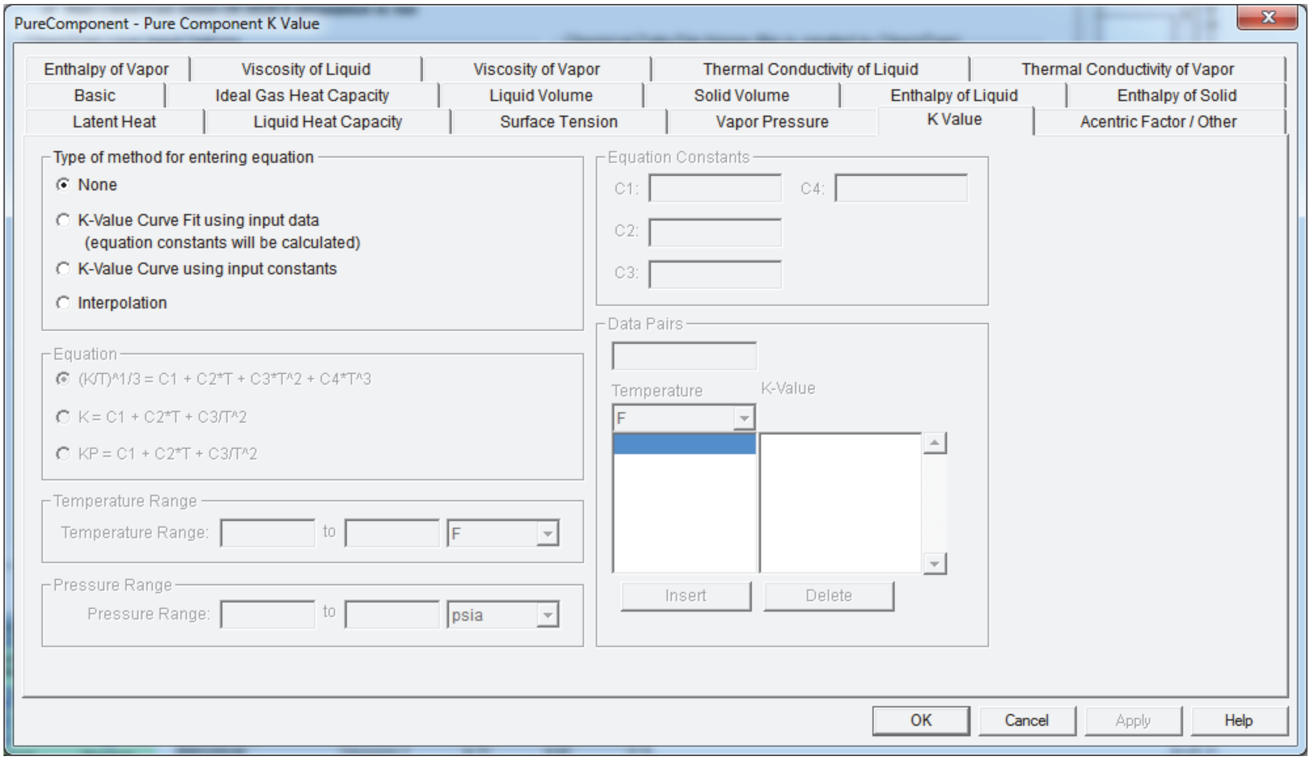


Figure 22: K Value tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can select the type of method to use for entering the K Value equation. You can enter K-values in the form of data, either for interpolation or curve fitting, or constants for one of three equation options. These equations give a functional dependence of K on T or K*P on T. Any K-value data entered overrides ChemTran or DESIGN II internal data systems.

Type of method for entering equation: Select one of the following:

None

K-Value Curve Fit using input data

K-Value Curve using input constants

Interpolation: the program will automatically select data for the appropriate temperature and pressure range for interpolation. If the ranges are not valid then an internal data correlation system is used. Linear interpolation is performed on data for an equation of the form ln K = f(1/T).

Equation Constants: If you selected K-Value Curve using input constants; then enter values in C1-C4.

Equation: If you selected either K-Value Curve choice, select the equation to use:

(K/T)^1/3 = C1 + C2*T + C3*T^2 + C4*T^3

K = C1 + C2*T + C3/T^2

KP = C1 + C2*T + C3/T^2

Temperature Range: If you selected any method choice except None, enter the lowest temperature limit value; then the highest and select a unit.

Pressure Range

If you selected Interpolation, enter the lowest pressure limitvalue; then the highest and select a unit.

Data Pairs: If you selected either the K-Value Curve Fit using input data or Interpolation choice. Click on the list box below the Temperature column name (underneath the unit drop down list); a blue highlight appears in the box. You can now type the desired value. Click the Insert button to add the value. Repeat this for the K-Value. You can create up to 50 data pairs. You can also select the units to use for Temperature and K-Value.

Acentric Factor/Other

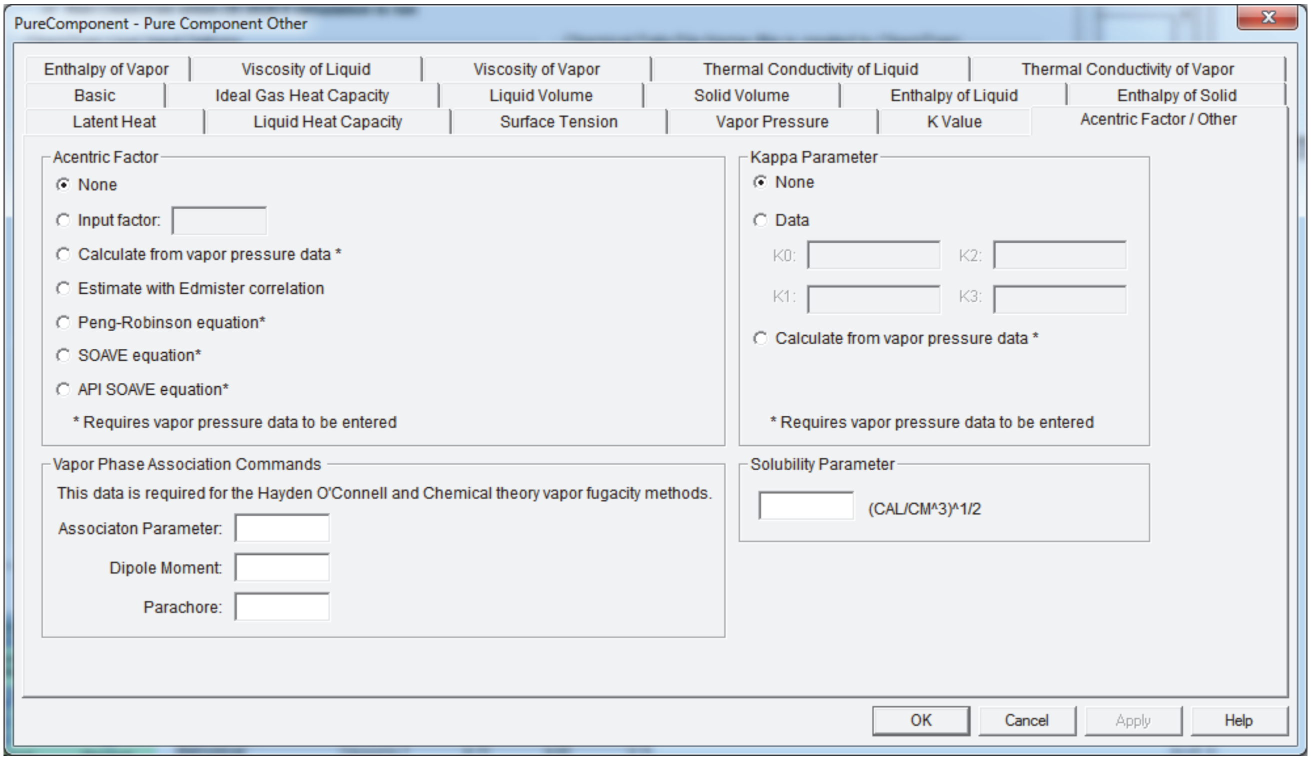


Figure 23: Acentric Factor/Other tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can select the type of method to use for entering the acentric factor, kappa parameter, vapor phase association commands, and solubility parameter. At any time, additional thermophysical data for a particular component may be provided directly to DESIGN II for simulation calculation. The provided data will be used rather than allowing DESIGN II to estimate properties for the components specified.

For the acentric factor, you can enter data, or request ChemTran to calculate or estimate it. If you choose none of these methods, ChemTran calculates the acentric factor from the vapor pressure/critical pressure/critical temperature, using the basic definition of the acentric factor by Pitzer.

For the Kappa Parameter, this is required when specifying the modified Peng-Robinson equation of state. If you choose none of the methods, the kappa parameters will default to the form of the standards Peng-Robinson equation.

Acentric Factor

Select one of the following:

- None
- Input factor: enter the factor
- Calculate from vapor pressure data*
- Estimate with Edmister correlation
- Peng-Robinson equation*
- SOAVE equation*
- API SOAVE equation*

* You must complete the Vapor Pressure tab.

Kappa Parameter

Select one of the following:

None

Data: enter values for K0, K1, K2, and K3

Calculate from vapor pressure data*: the Kappa parameters are fitted to the P-T data you entered for non-standard components, or using the vapor pressure curve from DESIGN II's library for standard components.

* You must complete the Vapor Pressure tab.

Vapor Phase Association Commands

This data is required for the Hayden O'Connell and Chemical theory vapor fugacitymethods. For the component, enter Association Parameter, Dipole Moment, and Parachore.

Solubility Parameter

Enter the solubility parameter using units of (CAL/CM^3)^1/2

Enthalpy of Vapor

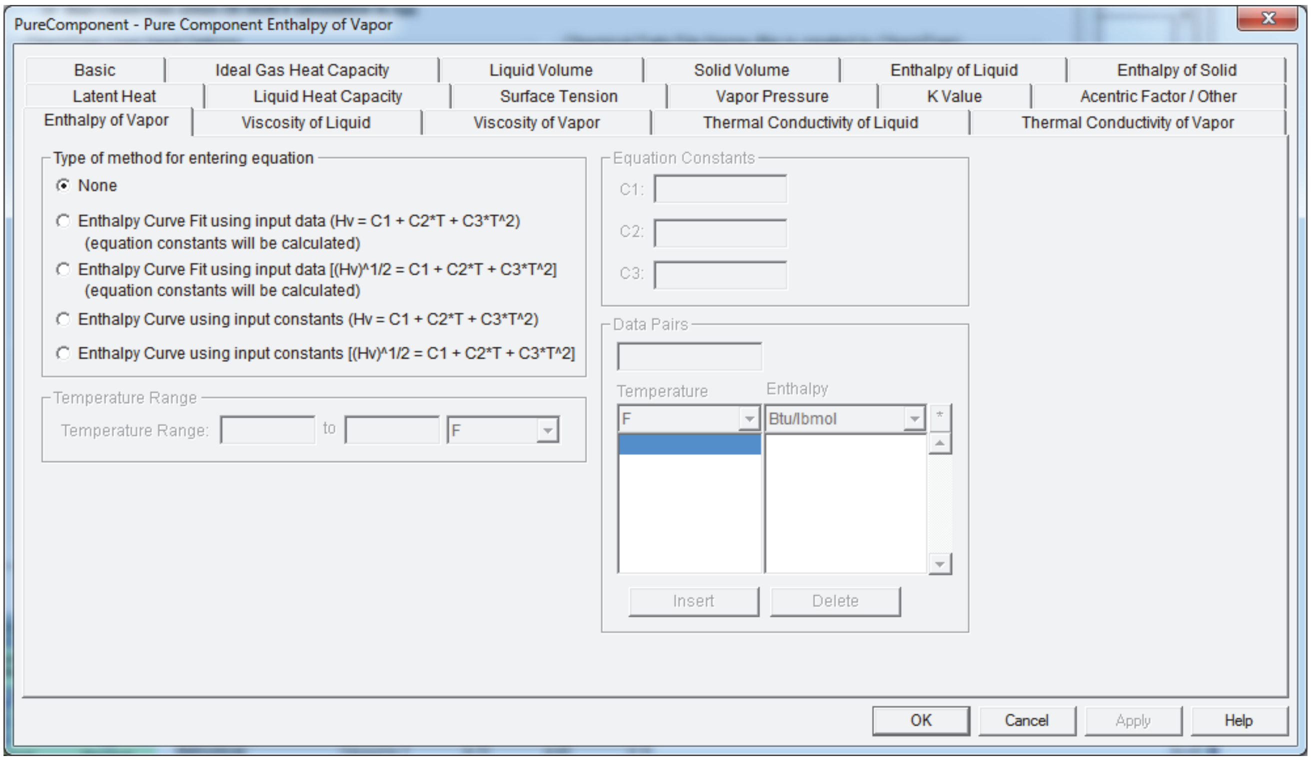


Figure 24: Enthalpy of Vapor tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can select the type of method to use for entering the Enthalpy of Vapor equation. You can enter vapor enthalpy versus temperature data for data reduction to one of two equations. Alternatively, you can enter the constants for the equation directly.

Type of method for entering equation: Select one of the following:

None

Enthalpy Curve Fit using input data (Hv = C1 + C2*T + C3*T^2)

Enthalpy Curve Fit using input data [(Hv)^1/2 = C1 + C2*T + C3*T^2]

Enthalpy Curve using input constants (Hv = C1 + C2*T + C3*T^2)

Enthalpy Curve using input constants [(Hv)^1/2 = C1 + C2*T + C3*T^2)

Equation Constants: If you selected either Enthalpy Curve using input constants choice; then enter values in C1-C3.

Temperature Range: If you selected any method other than None, enter the lowest temperature limit value; then the highest and select a unit.

Data Pairs: If you selected either Enthalpy Curve Fit using input data choice, click on the list box below the Temperature column name (underneath the unit drop down list); a blue highlight appears in the box. You can now type the desired value. Click the Insert button to add the value. Repeat this for the Enthalpy value. You can create up to 50 data pairs. You can also select the units to use for Temperature and Enthalpy.

Viscosity of Liquid

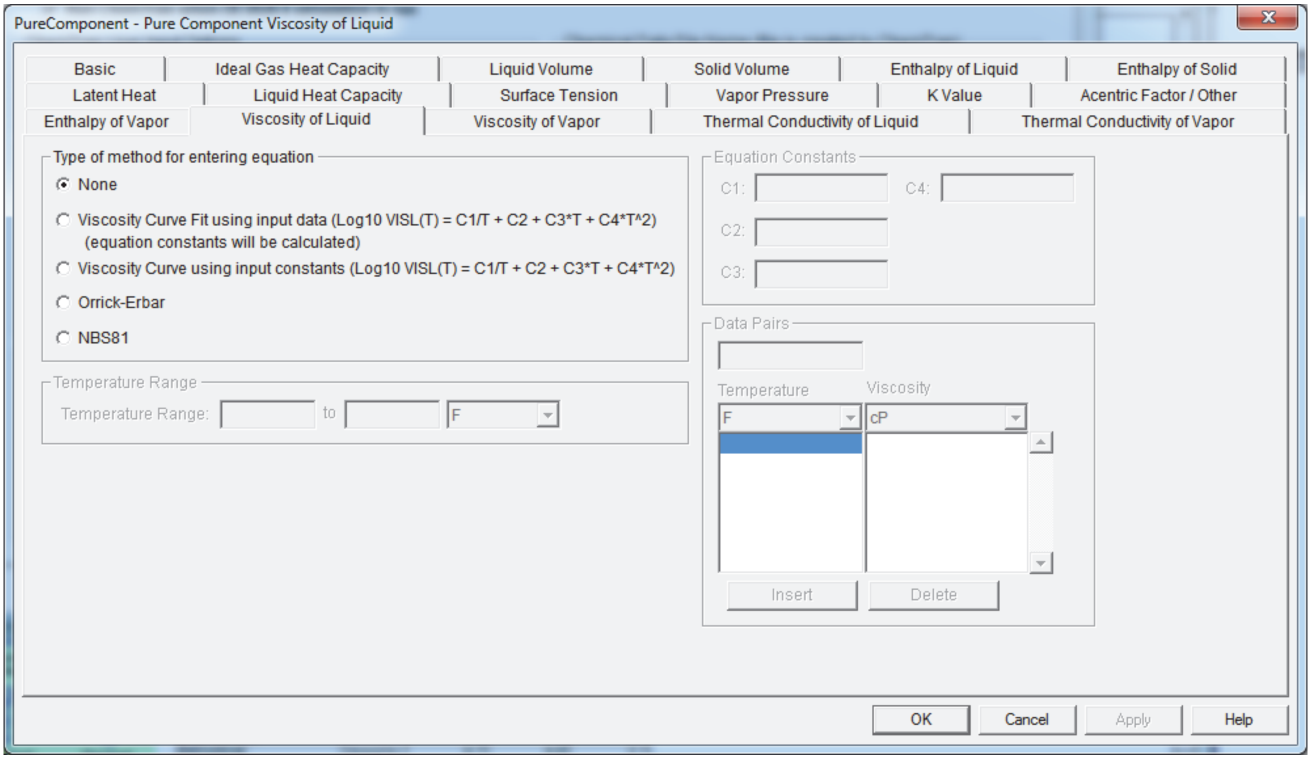


Figure 25: Viscosity of Liquid tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can select the type of method to use for entering the Viscosity of Liquid equation. You can enter data for liquid viscosity. Data can be fitted to a curve, correlation coefficients can be entered, or predictive methods may be specified.

Type of method for entering equation: Select one of the following:

None

Viscosity Curve Fit using input data (Log10 VISL(T) = C1/T + C2 + C3*T + C4*T^2)

Viscosity Curve using input constants (Log10 VISL(T) = C1/T + C2 + C3*T + C4*T^2)

Orrick-Erbar

NBS81

Equation Constants: If you selected Viscosity Curve using input constants; then enter values in C1-C4.

Temperature Range: If you selected either Viscosity Curve choice, enter the lowest temperature limit value; then the highest and select a unit.

Data Pairs: If you selected Viscosity Curve Fit using input data, click on the list box below the Temperature column name (underneath the unit drop down list); a blue highlight appears in the box. You can now type the desired value. Click the Insert button to add the value. Repeat this for the Viscosity value. You can create up to 50 data pairs. You can also select the units to use for Temperature and Viscosity.

Viscosity of Vapor

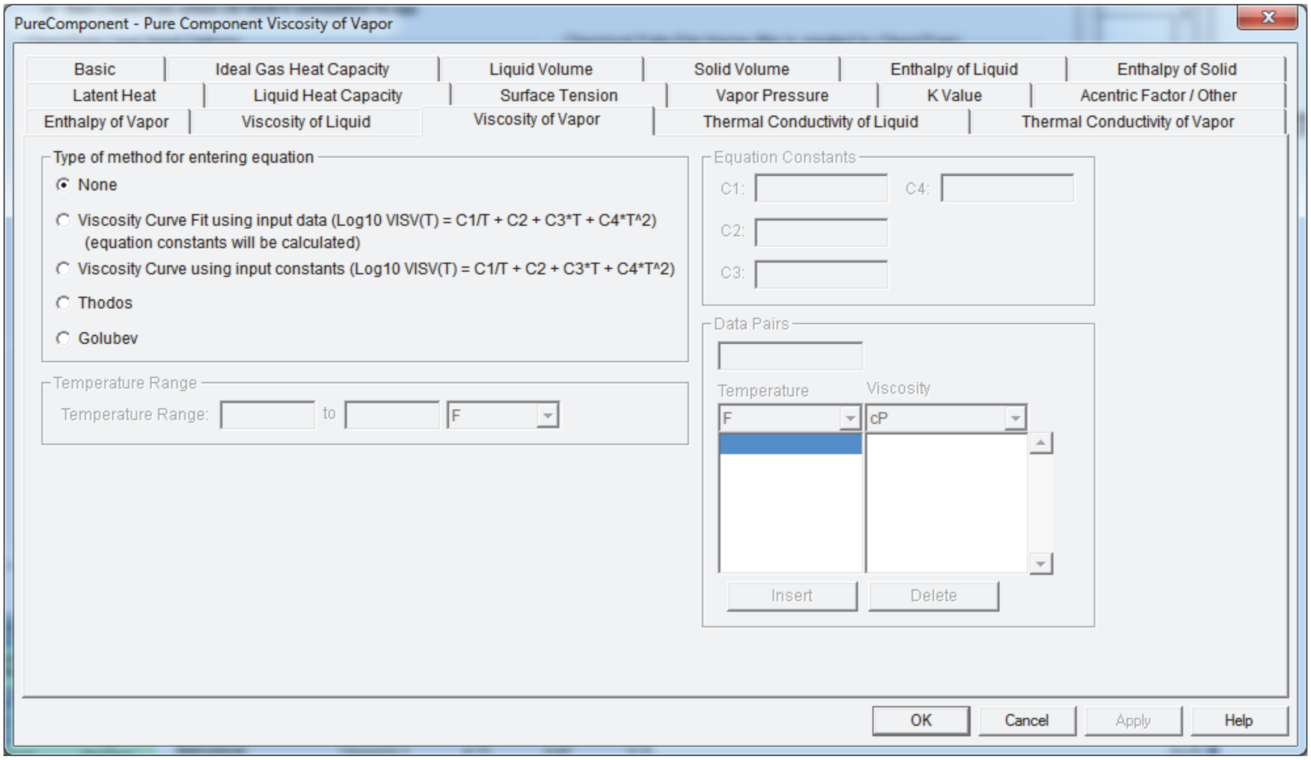


Figure 26: Viscosity of Vapor tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can select the type of method to use for entering the Viscosity of Vapor equation. You can enter data for vapor viscosity. Data can be fitted to a curve, correlation coefficients can be entered, or predictive methods may be specified.

Type of method for entering equation: Select one of the following:

None

Viscosity Curve Fit using input data (Log10 VISV(T) = C1/T + C2 + C3*T + C4*T^2)

Viscosity Curve using input constants (Log10 VISV(T) = C1/T + C2 + C3*T + C4*T^2)

Thodos

Golubev

Equation Constants: If you selected Viscosity Curve using input constants; then enter values in C1-C4.

Temperature Range: If you selected either Viscosity Curve choice, enter the lowest temperature limit value; then the highest and select a unit.

Data Pairs: If you selected Viscosity Curve Fit using input data, click on the list box below the Temperature column name (underneath the unit drop down list); a blue highlight appears in the box. You can now type the desired value. Click the Insert button to add the value. Repeat this for the Viscosity value. You can create up to 50 data pairs. You can also select the units to use for Temperature and Viscosity.

Thermal Conductivity of Liquid

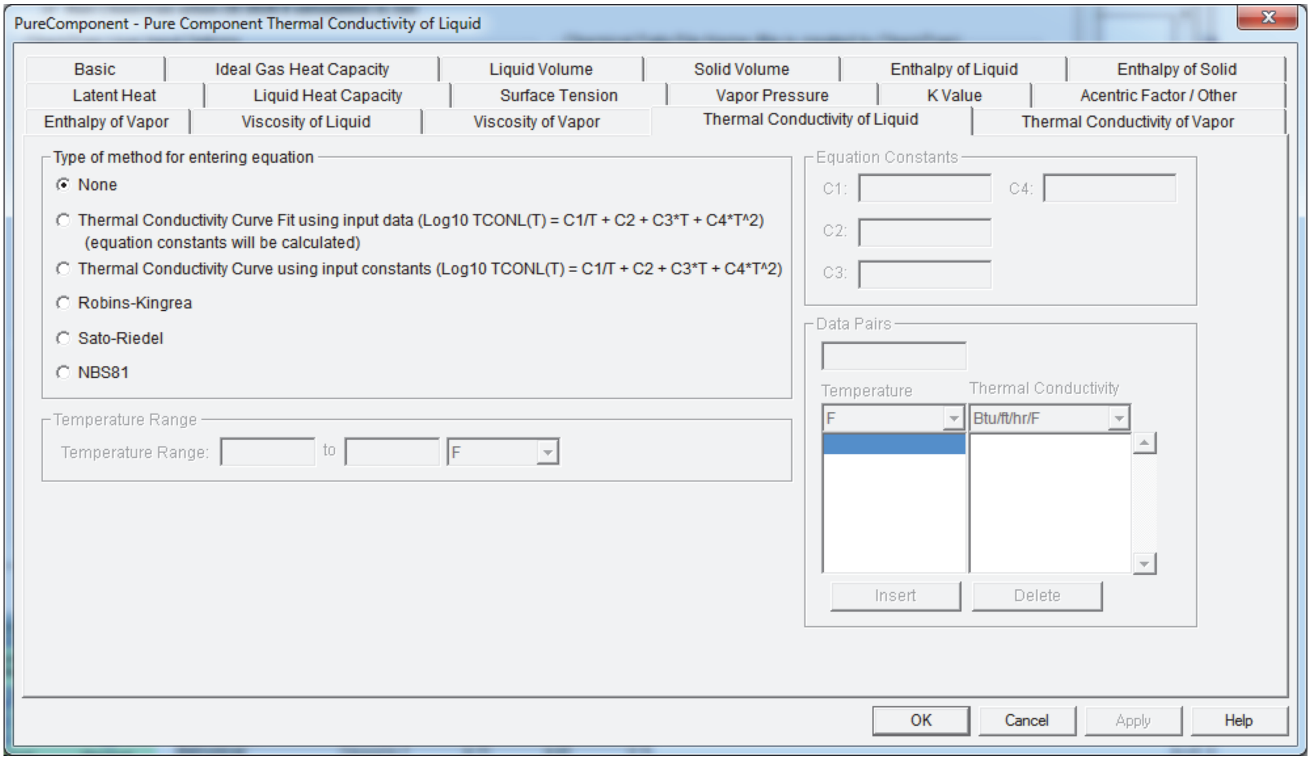


Figure 27: Thermal Conductivity of Liquid tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can select the type of method to use for entering the Thermal Conductivity of Liquid equation. You can enter data for liquid thermal conductivity. Data can be fitted to a curve, correlation coefficients can be entered, or predictive methods can be specified.

NOTE: You must select Ln Average for the Liquid Thermal Conductivity option under the Thermodynamic Methods tab on the Thermodynamic and Transport Methods dialog.

Type of method for entering equation: Select one of the following:

None

Thermal Conductivity Curve Fit using input data (Log10 TCONL(T) = C1/T + C2 + C3*T+ C4*T^2)

Thermal Conductivity Curve using input constants (Log10 TCONL(T) = C1/T + C2 + C3*T+ C4*T^2

Robins-Kingrea

Sato-Riedel

NBS81

Equation Constants: If you selected Thermal Conductivity Curve using input constants; then enter values in C1-C4.

Temperature Range: If you selected either Thermal Conductivity Curve choice, enter the lowest temperature limit value; then the highest and select a unit.

Data Pairs: If you selected Thermal Conductivity Curve Fit using input data, click on the list box below the Temperature column name (underneath the unit drop down list); a blue highlight appears in the box. You can now type the desired value. Click the Insert button to add the value. Repeat this for the Thermal Conductivity value. You can create up to 50 data pairs. You can also select the units to use for Temperature and Thermal Conductivity.

Thermal Conductivity of Vapor

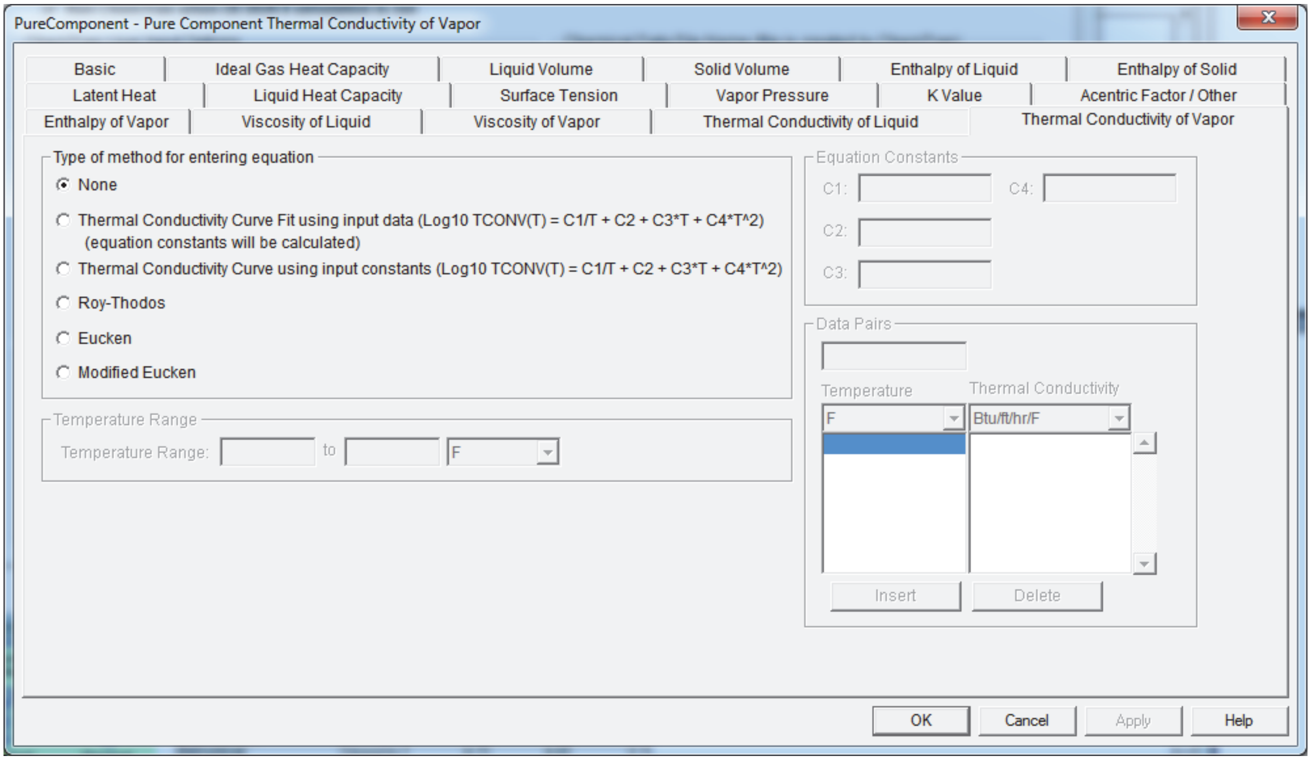


Figure 28: Thermal Conductivity of Vapor tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can select the type of method to use for entering the Thermal Conductivity of Vapor equation. You can enter data for vapor thermal conductivity. Data can be fitted to a curve, correlation coefficients can be entered, or predictive methods can be specified.

NOTE: You must select Ln Average for the Vapor Thermal Conductivity option under the Thermodynamic Methods tab on the Thermodynamic and Transport Methods dialog.

Type of method for entering equation: Select one of the following:

None

Thermal Conductivity Curve Fit using input data (Log10 TCONV(T) = C1/T + C2 + C3*T+ C4*T^2)

Thermal Conductivity Curve using input constants (Log10 TCONV(T) = C1/T + C2 + C3*T+ C4*T^2

Roy-Thodos

Eucken

Modified Eucken

Equation Constants: If you selected Thermal Conductivity Curve using input constants; then enter values in C1-C4.

Temperature Range: If you selected either Thermal Conductivity Curve choice, enter the lowest temperature limit value; then the highest and select a unit.

Data Pairs: If you selected Thermal Conductivity Curve Fit using input data, click on the list box below the Temperature column name (underneath the unit drop down list); a blue highlight appears in the box. You can now type the desired value.Click the Insert button to add the value. Repeat this for the Thermal Conductivity value. You can create up to 50 data pairs. You can also select the units to use for Temperature and Thermal Conductivity.

7. For the dialog input method, you can use the Mixture Commands button to display a dialog for creating and editing commands, copying existing ones, or deleting them. When you click the New or Edit button, the Mixture Commands dialog appears:

Mixture Commands

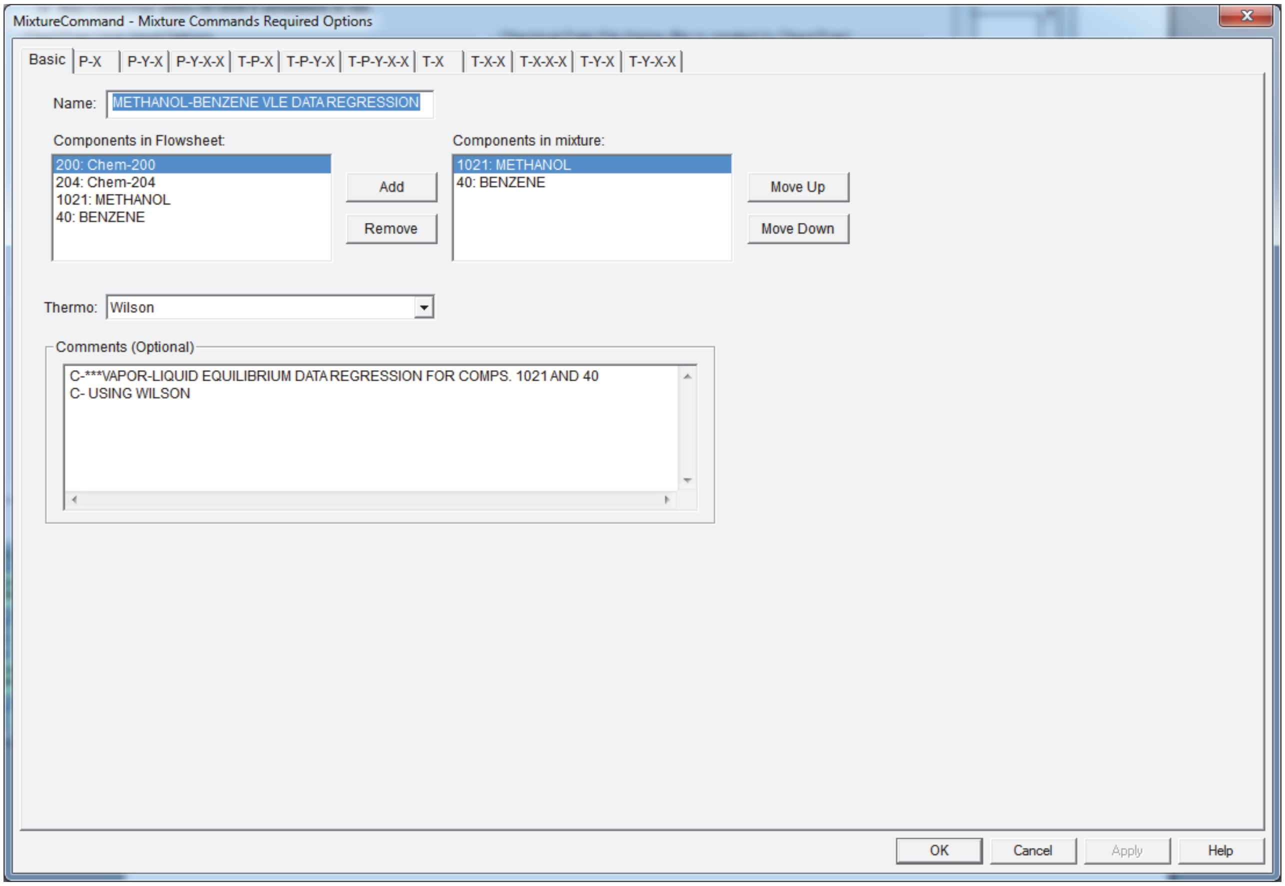


Figure 29: Pure Component Properties dialog (using ChemTran_PureComp & Mixture Dialog.psd)

8. Complete the following tabs as necessary:

Basic

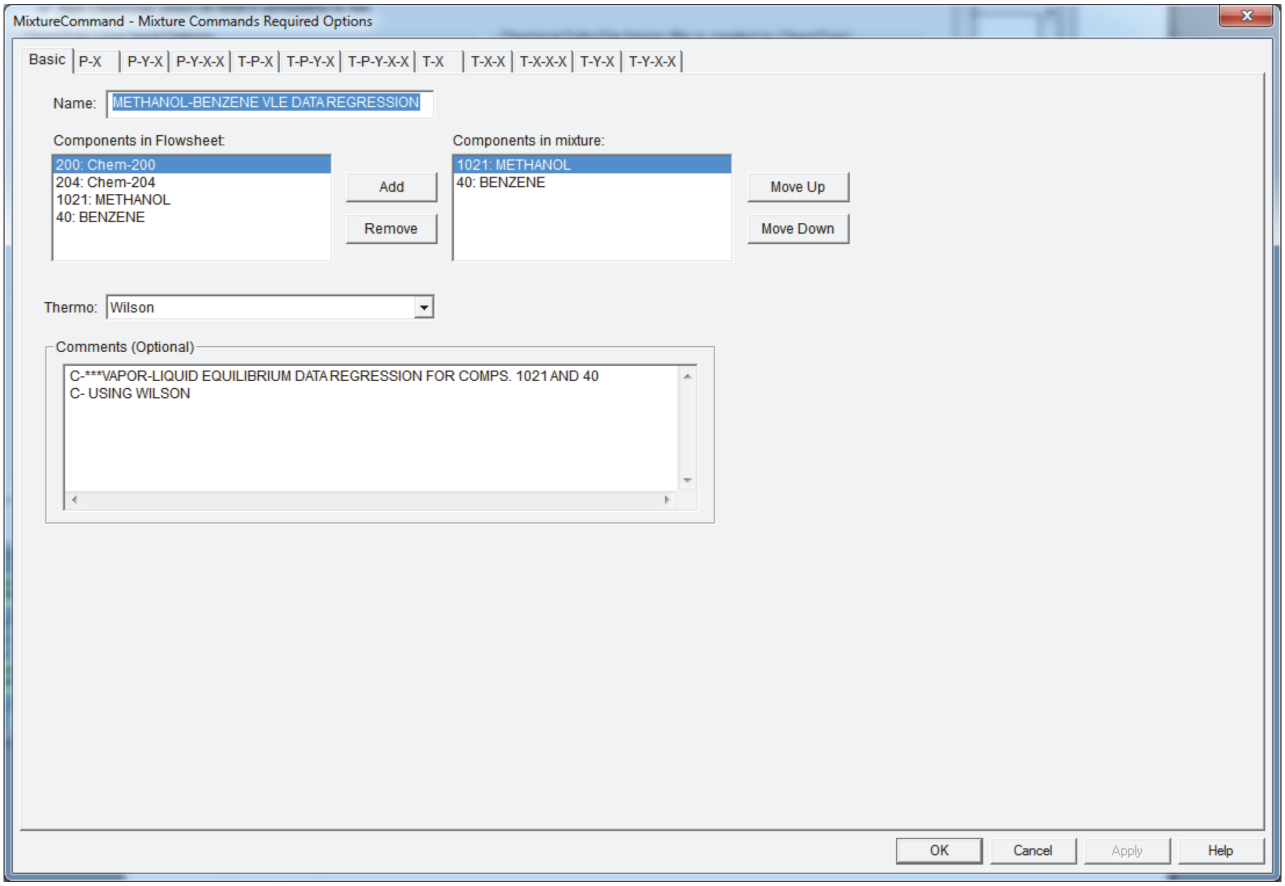


Figure 30: Basic tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can create a mixture command, select components for the mixture, and choose a thermodynamic method to use.

Name: Enter a name for the mixture command.

Components in Flowsheet: View the list of available components in the flowsheet. To select a component to add to the mixture, click on it then click Add (or simply double-click on the component).

Components in Mixture: View the list of components selected for the mixture. To remove a component from the list, click on it then click Remove. To change the order of a component in the list, click on it then click Move Up or Move Down.

Thermo: Select one of the following K-value correlation you want to use for data regression or equilibria prediction:

APISOAVE K

Ideal Vapor

Peng K

Renon

SOAVE K

Wilson

UNIQUAC K

UNIFAC K

Hayden O'Conell*

Chemical Theory Vapor*

* You must enter vapor phase association commands under the Acentric Factor/Other tab on the Pure Component Properties dialog.

Comments (Optional): You can enter any optional notes for the mixture command.

P-X

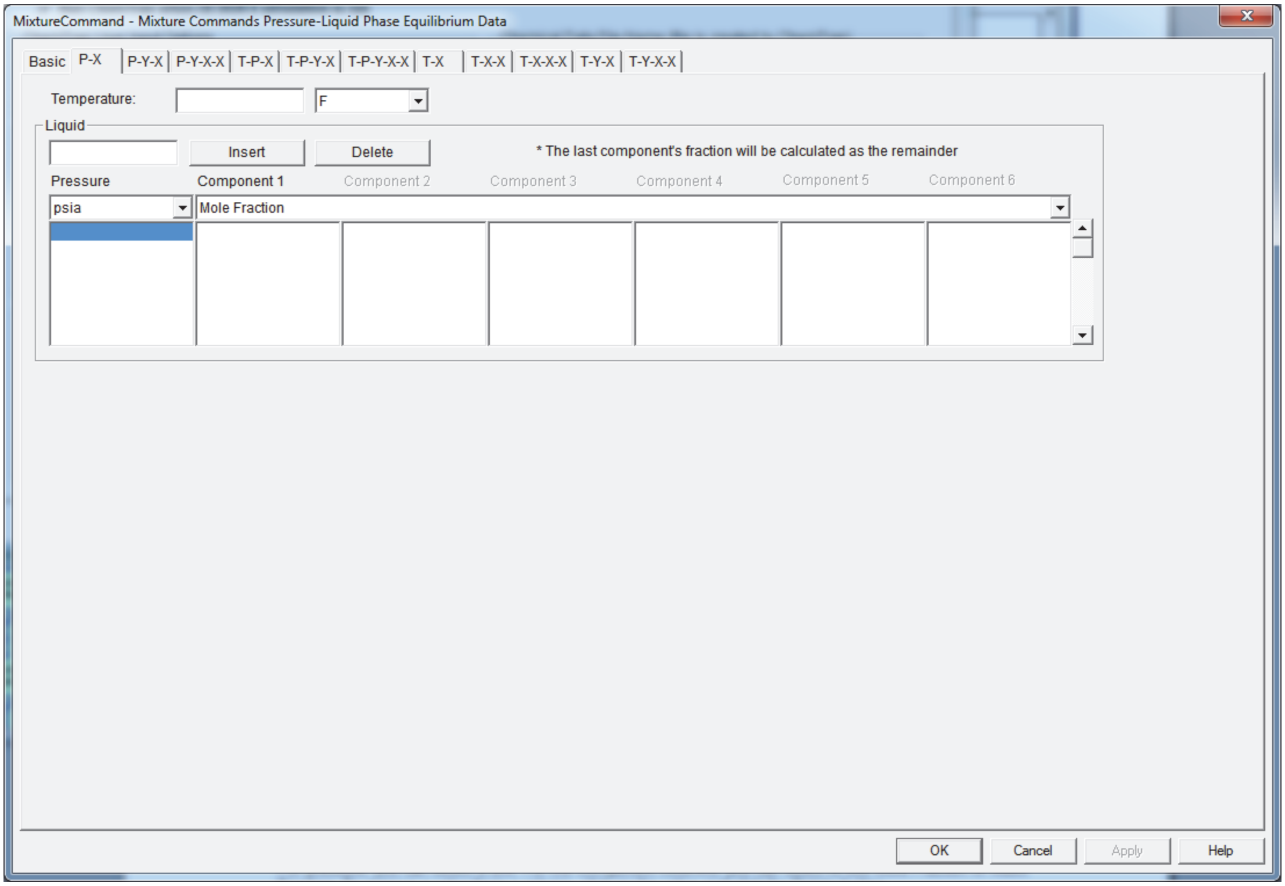


Figure 31: P-X tab (using ChemTran_PureComp & Mixture Dialog.psd)

For some experimental configurations, only T, P and liquid phase concentrations are measured even when the system consists of vapor and liquid in equilibrium. This type of data is usually referred to as solubility data; it is entered as P-X tab input using data collected at a constant temperature. Data collected at a constant pressure are entered using the T-X tab. If both T and P are permitted to vary; then the data is entered on the T-P-X tab.

P-X can only be used when the data represents liquid and vapor phases. The vapor phase concentrations will be accounted for by algorithms contained in ChemTran.

You can enter pressure versus liquid phase concentration at a constant temperature for a system composed of n components.

Temperature: Enter the constant temperature and select a unit.

Liquid: You will use this field to enter values under Pressure, Component 1 concentration, etc. Click on the list box below the appropriate column name (underneath the drop down list); a blue highlight appears in the box. You can now type the desired value (e.g. Pressure, Component 1 concentration, etc.). Click the Insert button to add the value. For pressure, you can select a unit. For the Component columns, you can select to use Mole Fraction or Mass Fraction. To remove a value from a column, click on the value and click the Delete button.

P-Y-X

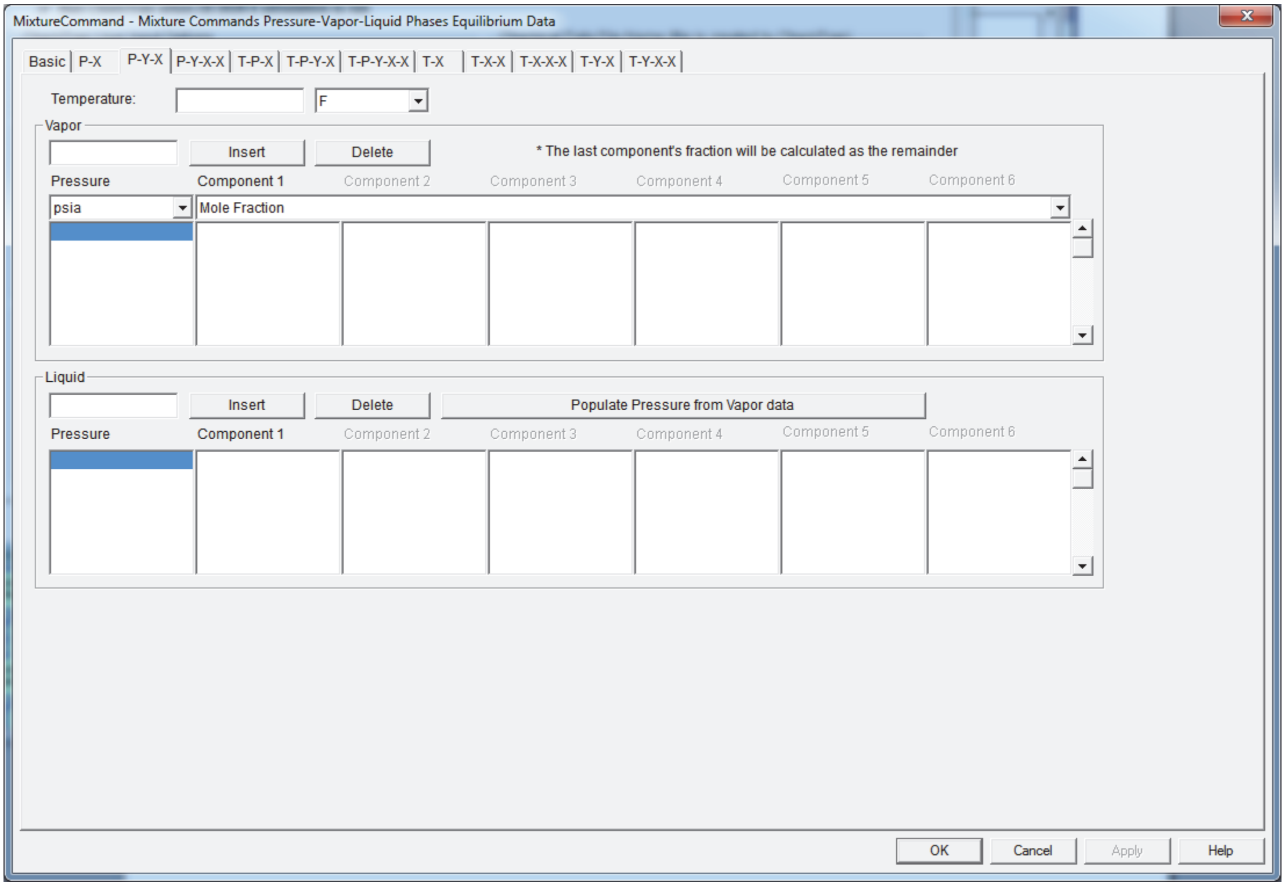


Figure 32: P-Y-X tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can enter V-L-E data where both the vapor and liquid phase concentrations are known, for a system composed of n components.

The P-Y-X tab is used for data taken at constant temperature. The T-Y-X tab is used for data taken at constant pressure. When both T and P vary, the T-P-Y-X tab is used.

Temperature: Enter the temperature and select a unit.

Vapor: You will use this field to enter values under Pressure, Component 1 concentration, etc. For pressure, you can select a unit. For the Component columns, you can select to use Mole Fraction or Mass Fraction.

Liquid: You will use this field to enter values under Pressure, Component 1 concentration, etc. To use the pressure values you entered for the vapor data, click the Populate button.

P-Y-X-X

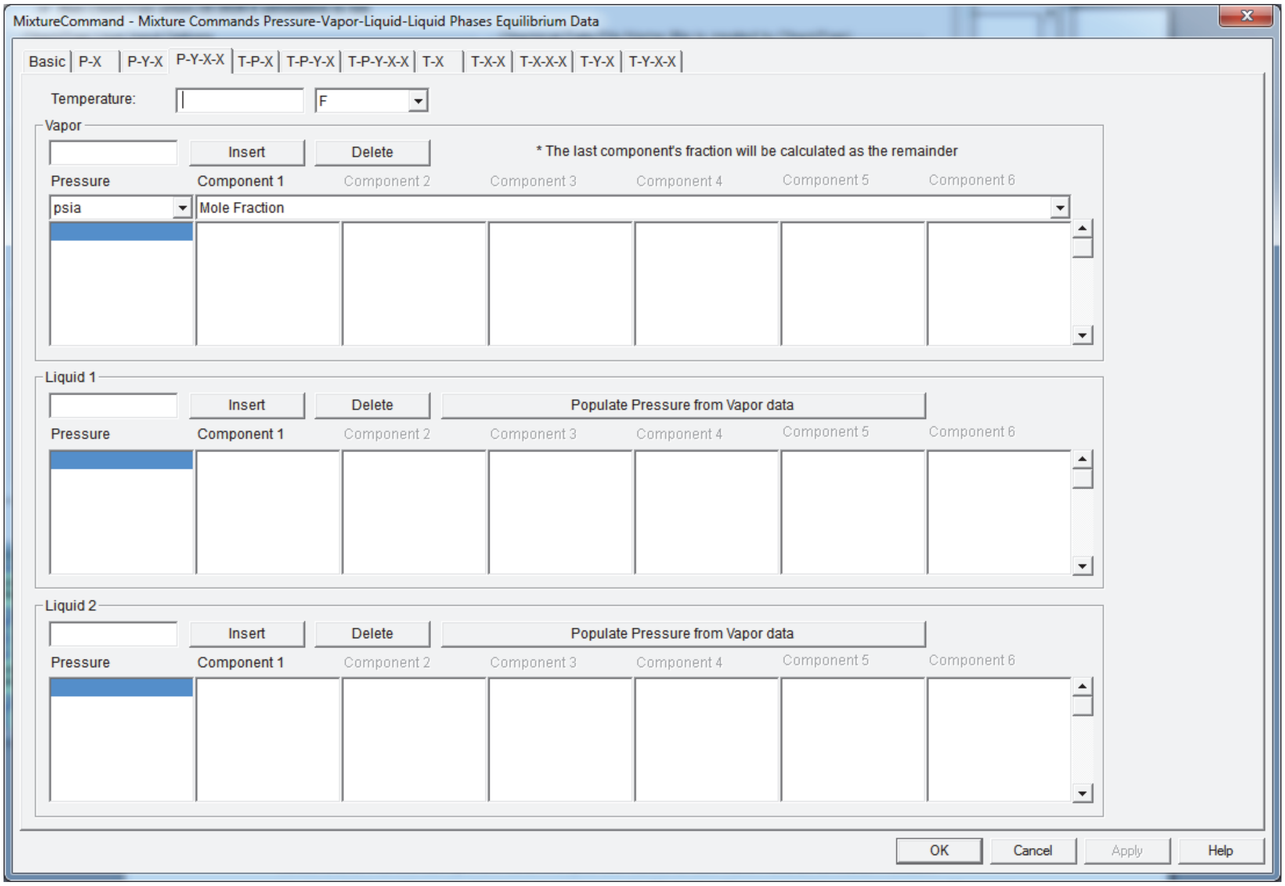


Figure 33: P-Y-X-X tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can enter vapor-liquid-liquid-equilibrium data, for a system composed of n components. This must be used with either the Renon or UNIQUAC thermo option set on the Basic tab. The P-Y-X-X tab is used for constant temperature data. The T-Y-X-X tab is used for constant pressure data. When both T and P vary, the T-P-Y-X-X tab can be used.

Temperature: Enter the temperature and select a unit.

Vapor: You will use this field to enter values under Pressure, Component 1 concentration, etc. For pressure, you can select a unit. For the Component columns, you can select to use Mole Fraction or Mass Fraction.

Liquid 1, 2: You will use this field to enter values under Pressure, Component 1 concentration, etc. To use the pressure values you entered for the vapor data, click the Populate button.

T-P-X

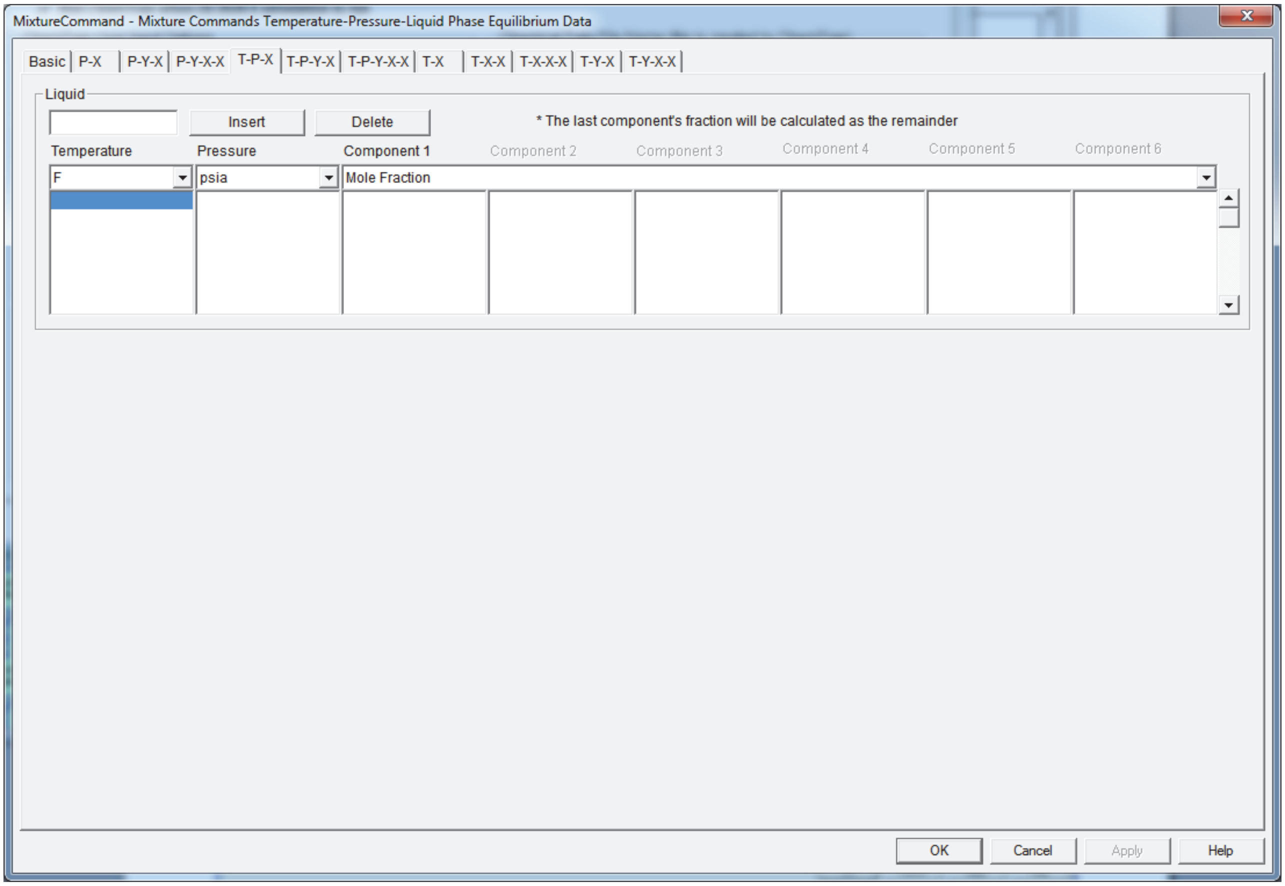


Figure 34: T-P-X tab (using ChemTran_PureComp & Mixture Dialog.psd)

For some experimental configurations, only T, P and liquid phase concentrations are measured even when the system consists of vapor and liquid in equilibrium. This type of data is usually referred to as solubility data; it is entered as T-P-X tab input using varying pressure and temperature data. Data collected at a constant temperature are entered using the P-X tab. Data collected at a constant pressure are entered using the T-X tab.

T-P-X can only be used when the data represents liquid and vapor phases. The vapor phase concentrations will be accounted for by algorithms contained in ChemTran.

You can enter varying temperature and pressure versus liquid phase concentration for a system composed of n components.

Liquid: You will use this field to enter values under Temperature, Pressure, Component 1 concentration, etc. For temperature and pressure, you can select a unit. For the Component columns, you can select to use Mole Fraction or Mass Fraction.

T-P-Y-X

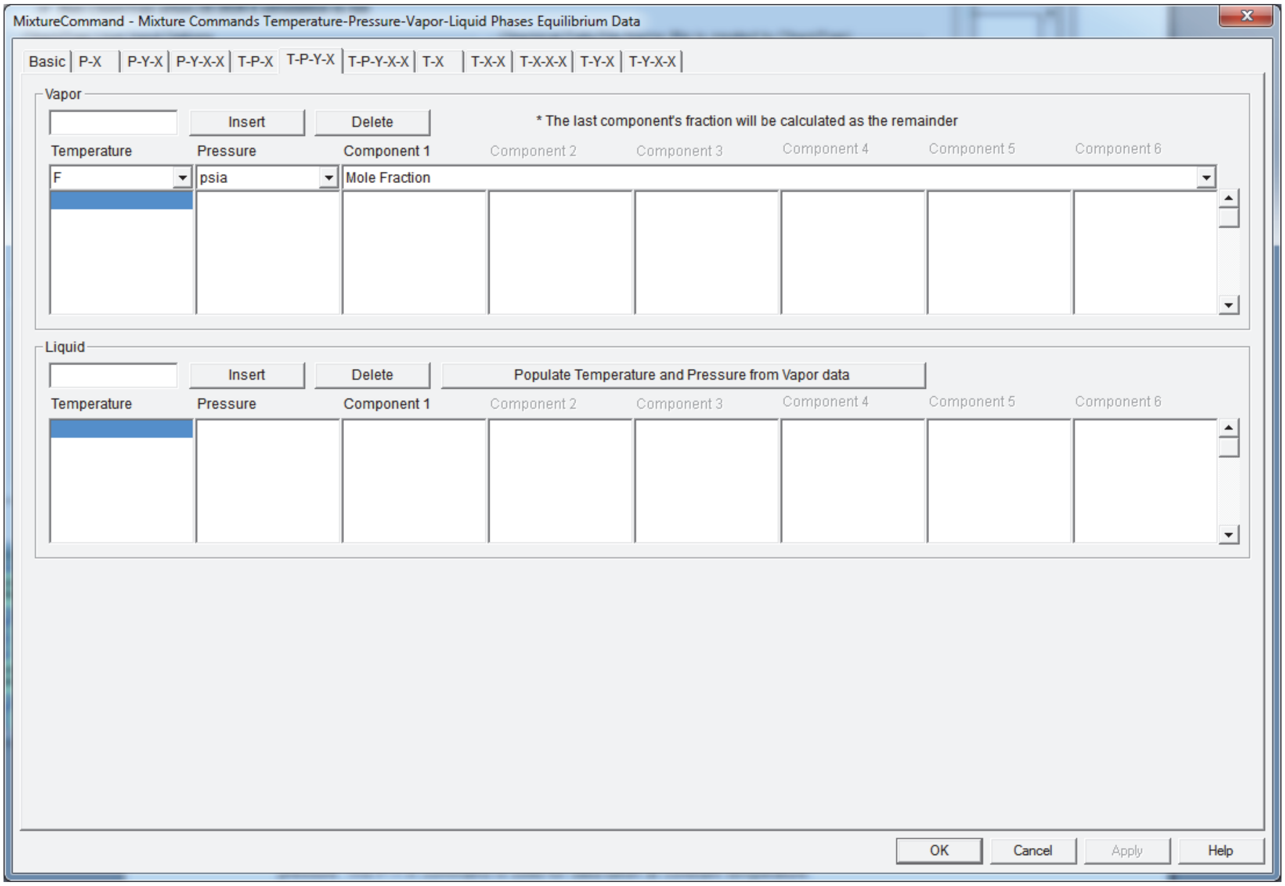


Figure 35: T-P-Y-X tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can enter V-L-E data where both the vapor and liquid phase concentrations are known, for a system composed of n components. When both T and P vary, the T-P-Y-X tab is used. The T-Y-X tab is used for data taken at constant pressure. The P-Y-X command is used for data taken at constant temperature.

Vapor: You will use this field to enter values under Temperature, Pressure, Component 1 concentration, etc. For temperature and pressure, you can select a unit. For the Component columns, you can select to use Mole Fraction or Mass Fraction.

Liquid: You will use this field to enter values under Temperature, Pressure, Component 1 concentration, etc. To use the temperature and pressure values you entered for the vapor data, click the Populate button.

T-P-Y-X-X

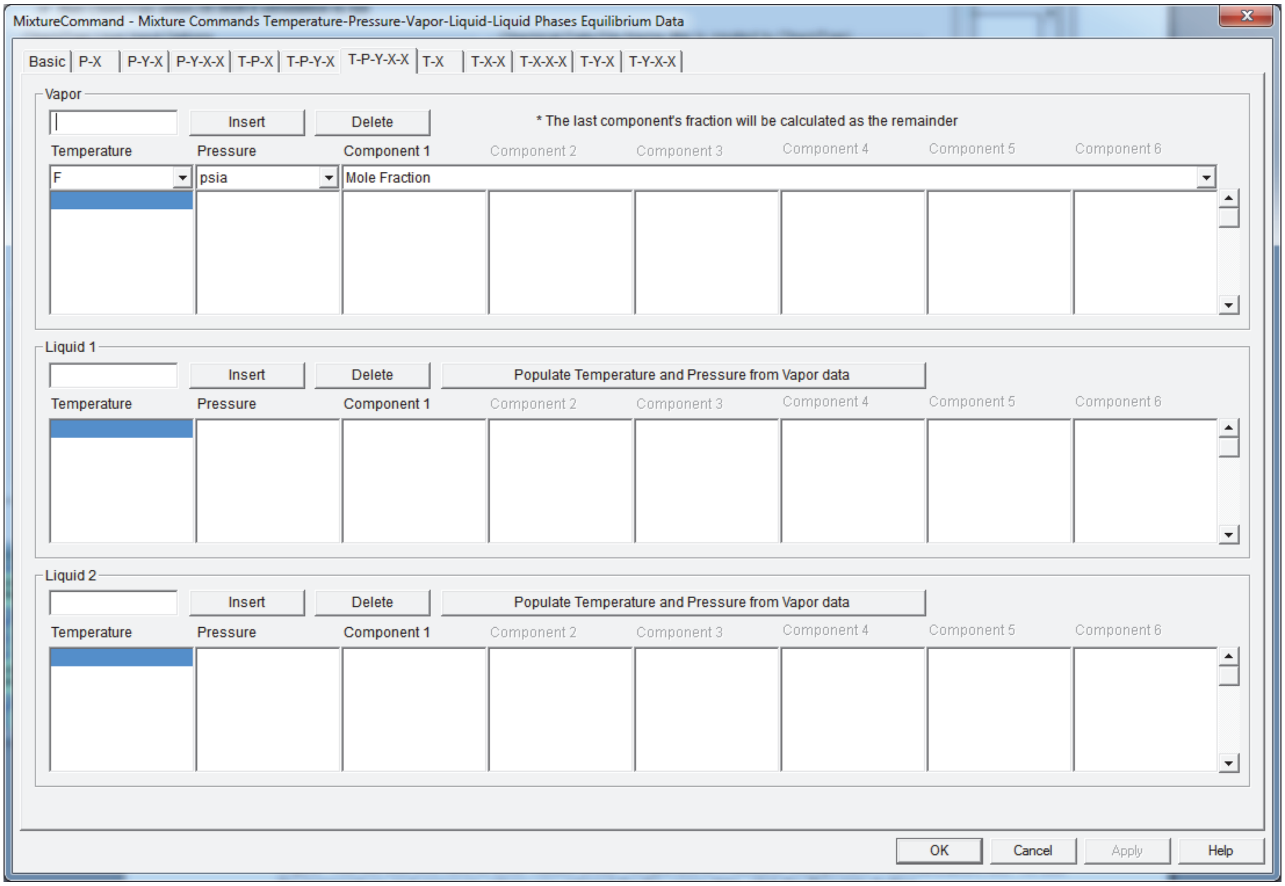


Figure 36: T-P-Y-X-X tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can enter vapor-liquid-liquid-equilibrium data, for a system composed of n components. This must be used with either the Renon or UNIQUAC thermo option set on the Basic tab. When both T and P vary, the T-P-Y-X-X tab is used. The P-Y-X-X tab is used for constant temperature data. The T-Y-X-X tab is used for constant pressure data.

Vapor: You will use this field to enter values under Temperature, Pressure, Component 1 concentration, etc. For temperature and pressure, you can select a unit. For the Component columns, you can select to use Mole Fraction or Mass Fraction.

Liquid 1, 2: You will use this field to enter values under Temperature, Pressure, Component 1 concentration, etc. To use the temperature and pressure values you entered for the vapor data, click the Populate button.

T-X

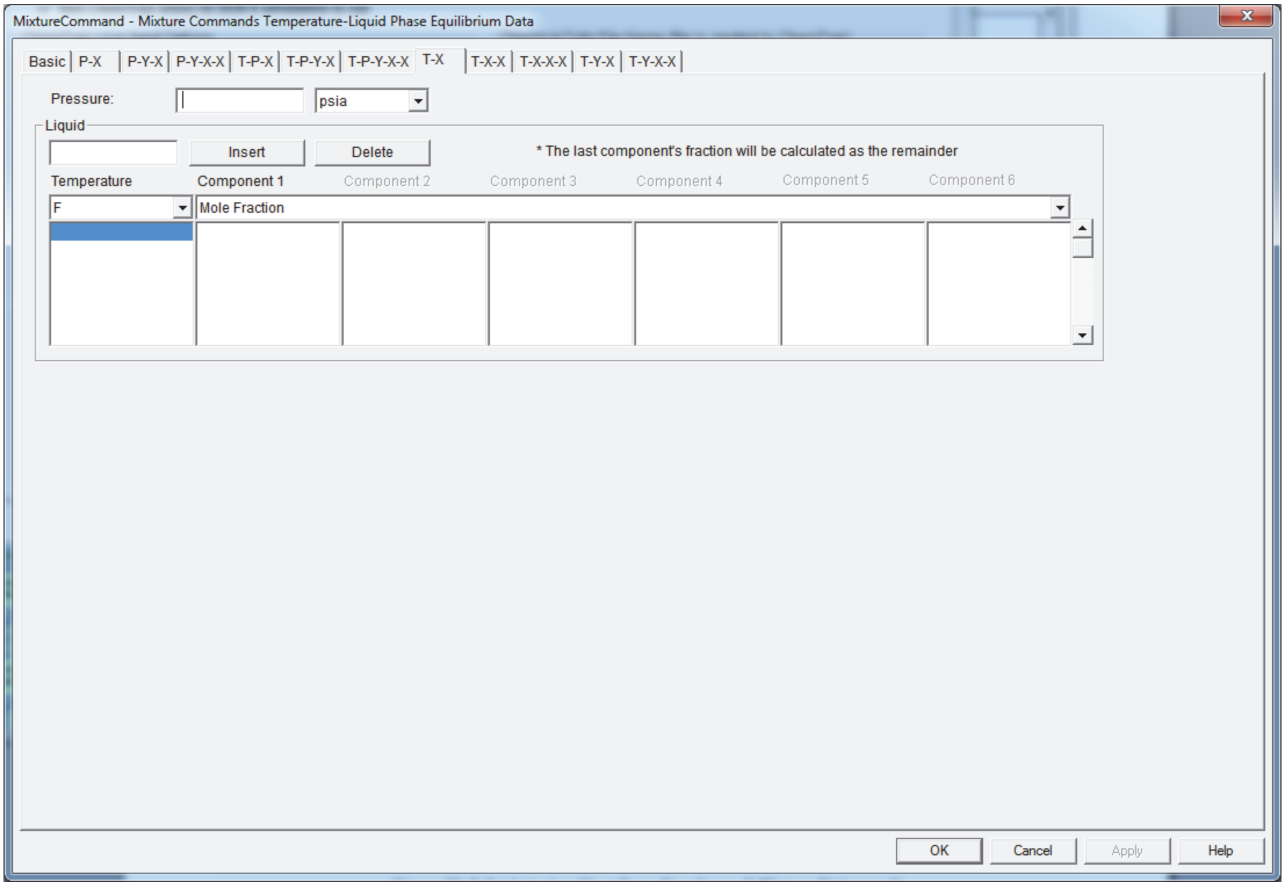


Figure 37: T-X tab (using ChemTran_PureComp & Mixture Dialog.psd)

For some experimental configurations, only T, P and liquid phase concentrations are measured even when the system consists of vapor and liquid in equilibrium. This type of data is usually referred to as solubility data; it is entered as T-X tab input using data collected at a constant pressure. Data collected at a constant temperature are entered using the P-X tab. If both T and P are permitted to vary; then the data is entered on the T-P-X tab.

T-X can only be used when the data represents liquid and vapor phases. The vapor phase concentrations will be accounted for by algorithms contained in ChemTran. You can use this tab on the Mixture Command Properties dialog to enter temperature versus liquid phase concentration at a constant pressure for a system composed of n components.

Pressure: Enter the constant pressureand select a unit.

Liquid: You will use this field to enter values under Temperature, Component 1 concentration, etc. For temperature, you can select a unit. For the Component columns, you can select to use Mole Fraction or Mass Fraction.

T-X-X

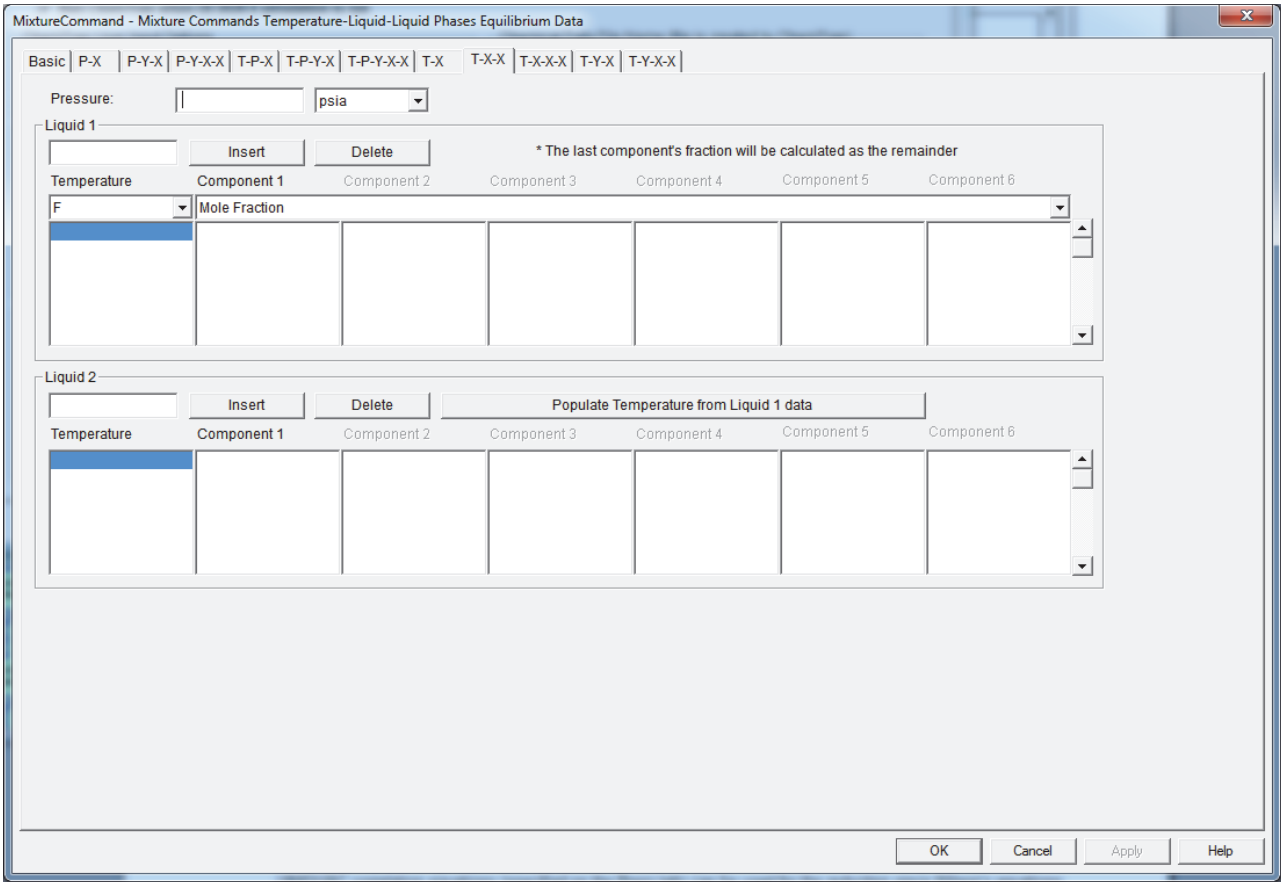


Figure 38: T-X-X tab (using ChemTran_PureComp & Mixture Dialog.psd)

Enter data for partially miscible liquid-liquid systems using this tab on the Mixture Command Properties dialog. For these systems, the temperature and concentrations of both liquid phases must be entered. Only the Renon and UNIQUAC correlation equations (specified on the Basic tab) can be used for the reduction since Wilson's equations do not account for immiscibility.

Pressure: Enter the constant pressure and select a unit.

Liquid 1: You will use this field to enter values under Temperature, Component 1 concentration, etc, For temperature, you can select a unit. For the Component columns, you can select to use Mole Fraction or Mass Fraction.

Liquid 2: You will use this field to enter values under Temperature, Component 1 concentration, etc. To use the temperature values you entered for the Liquid 1 data, click the Populate button.

T-X-X-X

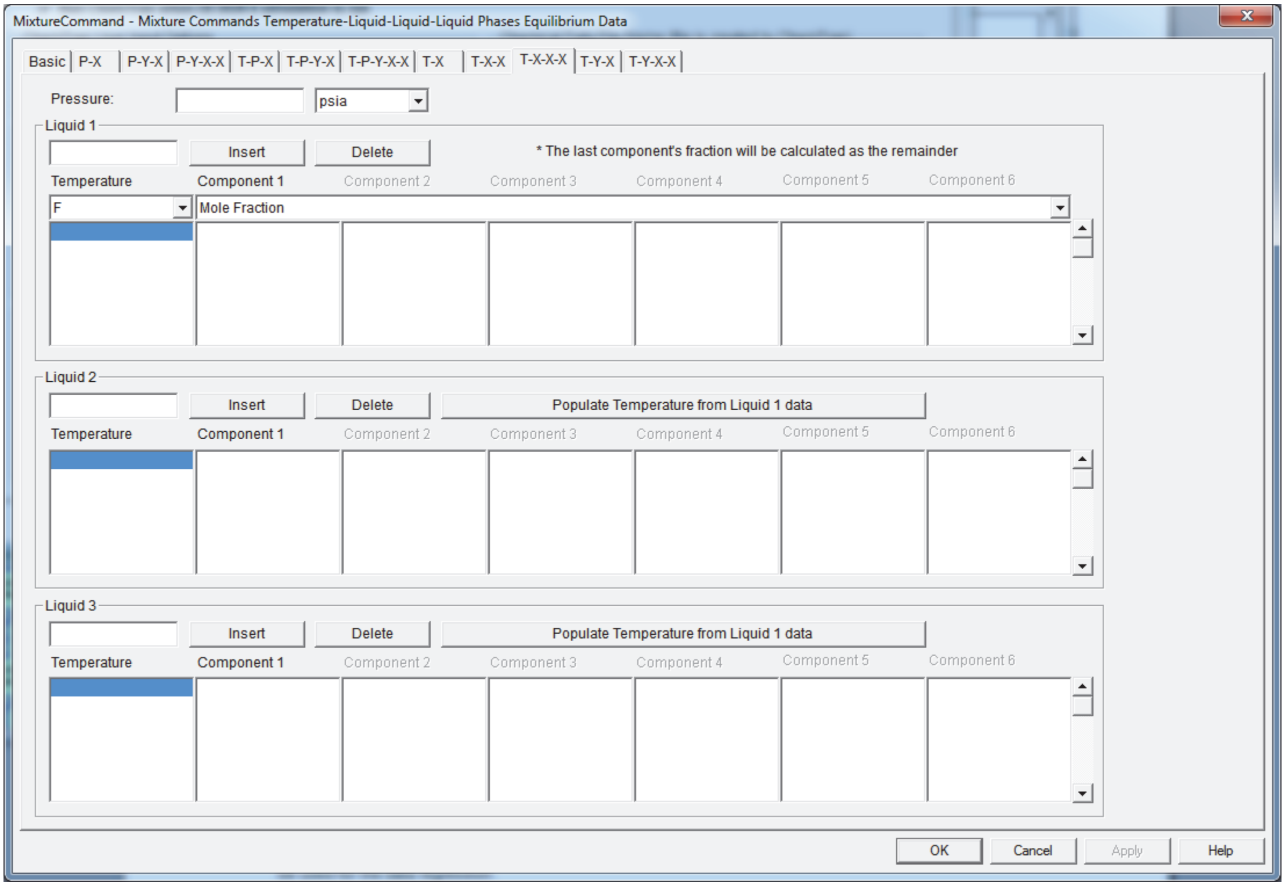


Figure 39: T-X-X-X tab (using ChemTran_PureComp & Mixture Dialog.psd)

Enter mixture data on systems with three liquid phases in equilibrium. The temperature and compositions for all three liquid phases must be entered. Only the Renon and UNIQUAC correlation equations (specified on the Basic tab) can be used for the data regression.

Pressure: Enter the constant pressure and select a unit.

Liquid 1: You will use this field to enter values under Temperature, Component 1 concentration, etc. For temperature, you can select a unit. For the Component columns, you can select to use Mole Fraction or Mass Fraction.

Liquid 2: You will use this field to enter values under Temperature, Component 1 concentration, etc. To use the temperature values you entered for the Liquid 1 data, click the Populate button.

Liquid 3: You will use this field to enter values under Temperature, Component 1 concentration, etc. To use the temperature values you entered for the Liquid 1 data, click the Populate button.

T-Y-X

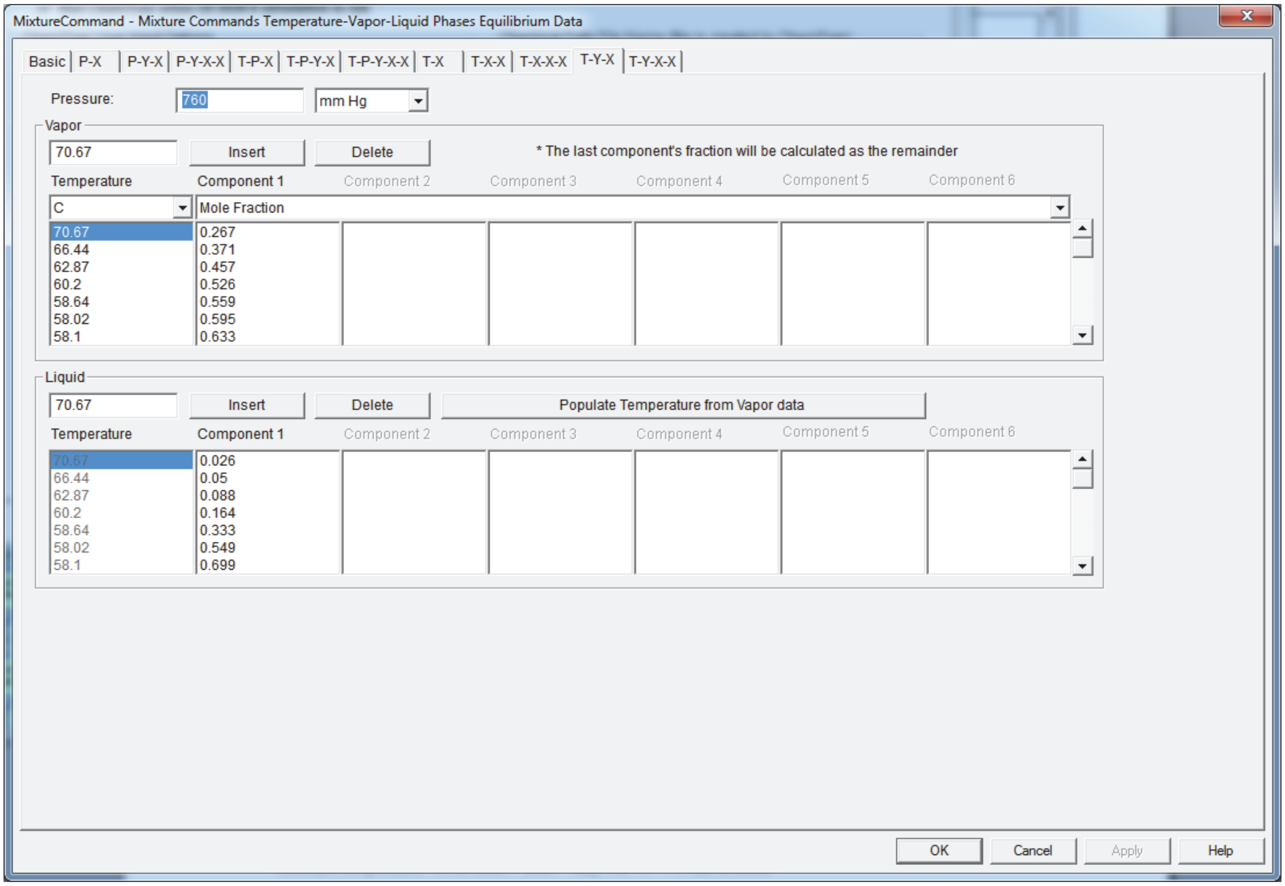


Figure 40: T-Y-X tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can enter V-L-E data where both the vapor and liquid phase concentrations are known, for a system composed of n components. The T-Y-X tab is used for data taken at constant pressure. The P-Y-X tab is used for data taken at constant temperature. When both T and P vary, the T-P-Y-X tab is used.

Pressure: Enter the pressure and select a unit.

Vapor: You will use this field to enter values under Temperature, Component 1 concentration, etc. For temperature, you can select a unit. For the Component columns, you can select to use Mole Fraction or Mass Fraction.

Liquid: You will use this field to enter values under Temperature, Component 1 concentration, etc. To use the temperature values you entered for the vapor data, click the Populate button.

T-Y-X-X

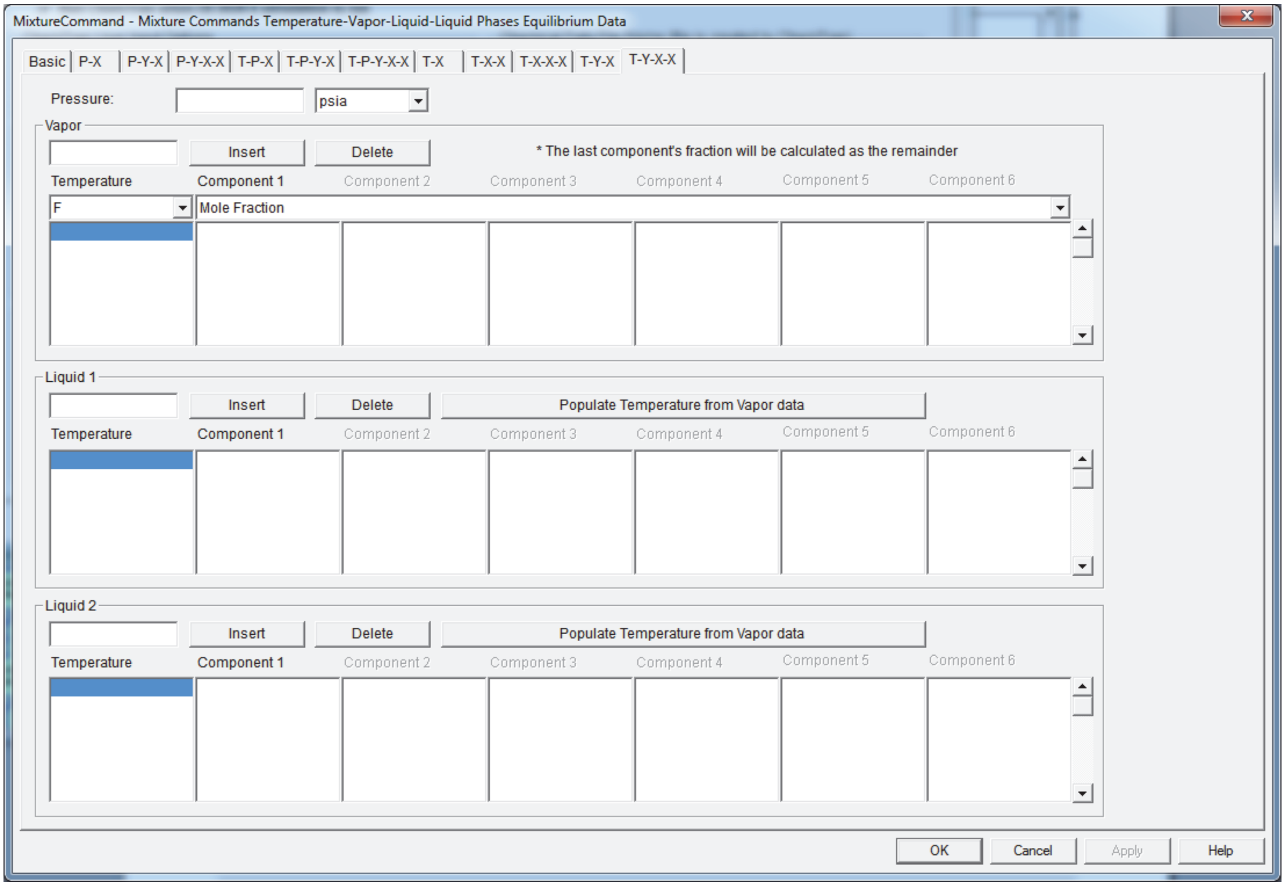


Figure 41: T-Y-X-X tab (using ChemTran_PureComp & Mixture Dialog.psd)

You can enter vapor-liquid-liquid-equilibrium data, for a system composed of n components. This must be used with either the Renon or UNIQUAC thermo option set on the Basic tab. The T-Y-X-X tab is used for constant pressure data. The P-Y-X-X tab is used for constant pressure data. When both T and P vary, the T-P-Y-X-X tab can be used.

Temperature: Enter the temperature and select a unit.

Vapor: You will use this field to enter values under Pressure, Component 1 concentration, etc. For pressure, you can select a unit. For the Component columns, you can select to use Mole Fraction or Mass Fraction.

Liquid 1, 2: You will use this field to enter values under Pressure, Component 1 concentration, etc. To use the temperature values you entered for the vapor data, click the Populate button.

Component Data

You can enter Critical Temperature, Critical Pressure, Critical Volume, and Acentric Factor data for components 100 to 199 and 500 to 999 only.

Open the **Specify** menu and select **Components**. The Components dialog appears with the Components tab selected. Click on the desired tab: Component General Properties, Component Critical Properties, or Component Heat of Reaction Properties. **Note:** If you are using Crude feed commands to describe any feeds in your flowsheet, you cannot use the petroleum fraction commands.

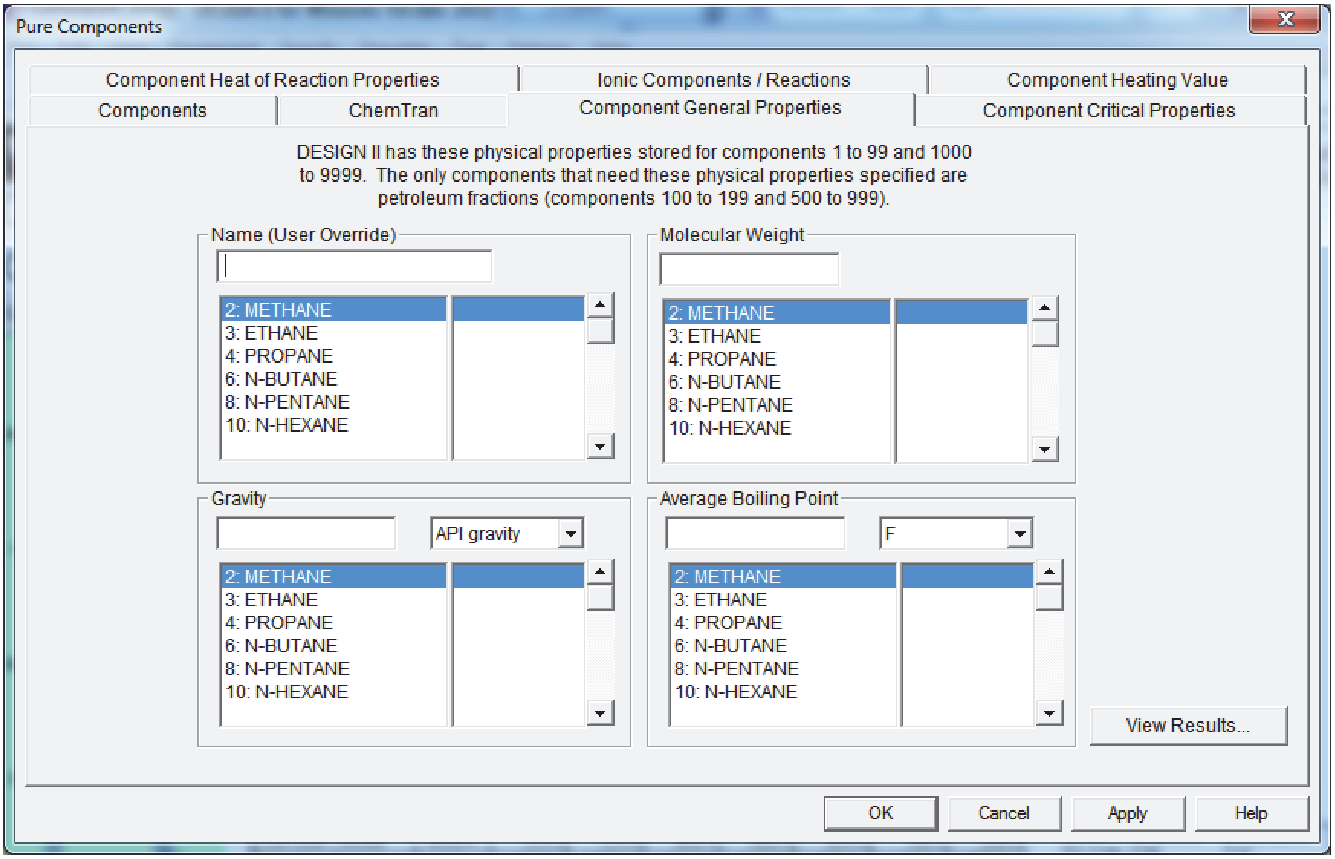


Figure 42: Component General Properties (from expander.psd)

1. Component General Properties tab: Enter general properties for any petroleum fractions, components between 100-199 and 500 to 999. You must set a Gravity and Average Boiling Point for each petroleum fraction. You can also optionally enter a Molecular Weight (estimated from AMB and API if you do not any one) and/or a new name to use for the component (if no name is entered, the fraction will be identified by its specified boiling point). For Name, the maximum is 16 characters; do not use commas in the name.

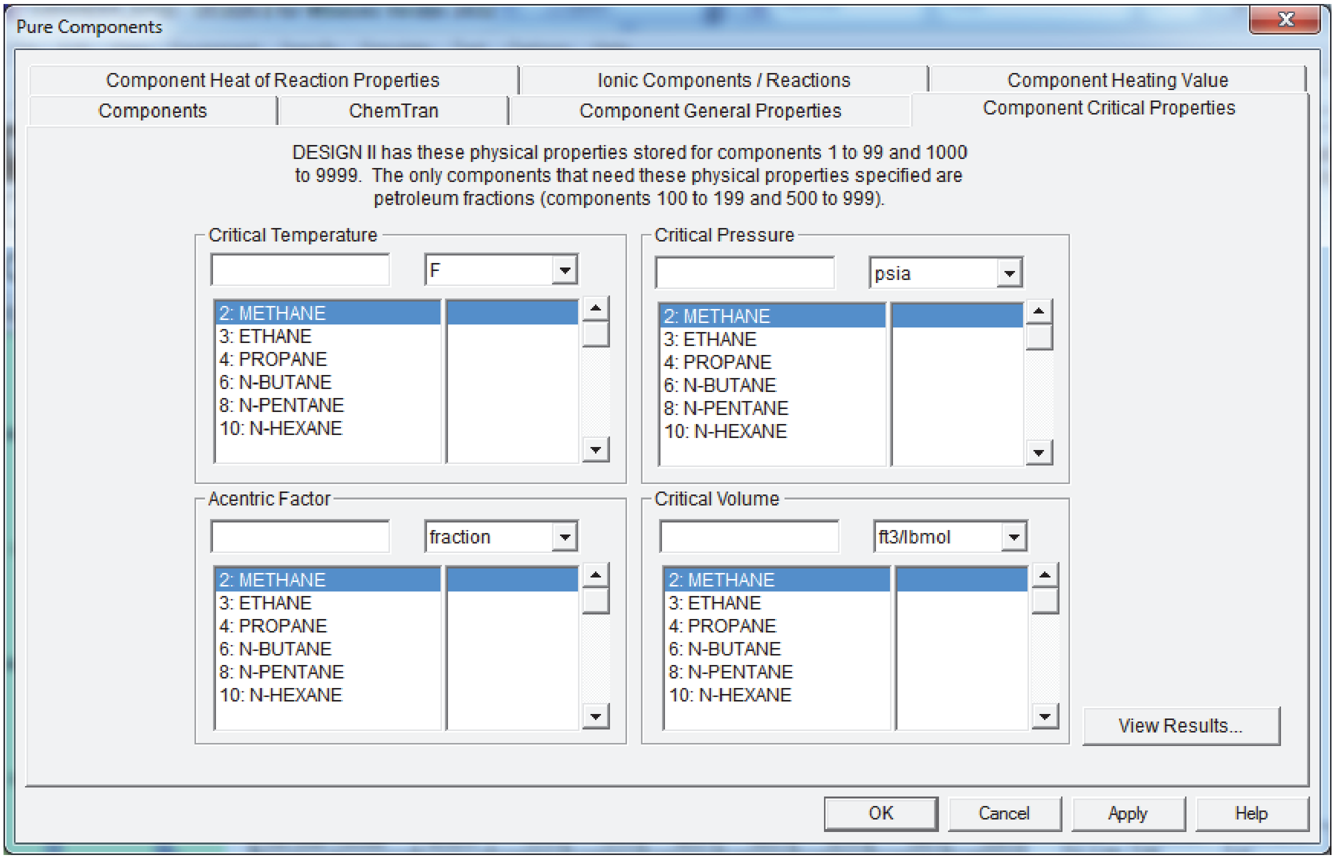


Figure 43: Component Critical Properties

Component Critical Properties tab: Enter critical properties for any petroleum fractions, components between 100-199 and 500 to 999. You can adjust the calculation of thermophysical properties for pseudocomponents. Enter a Critical Temperature, Critical Pressure, Acentric Factor, and Critical Volume.

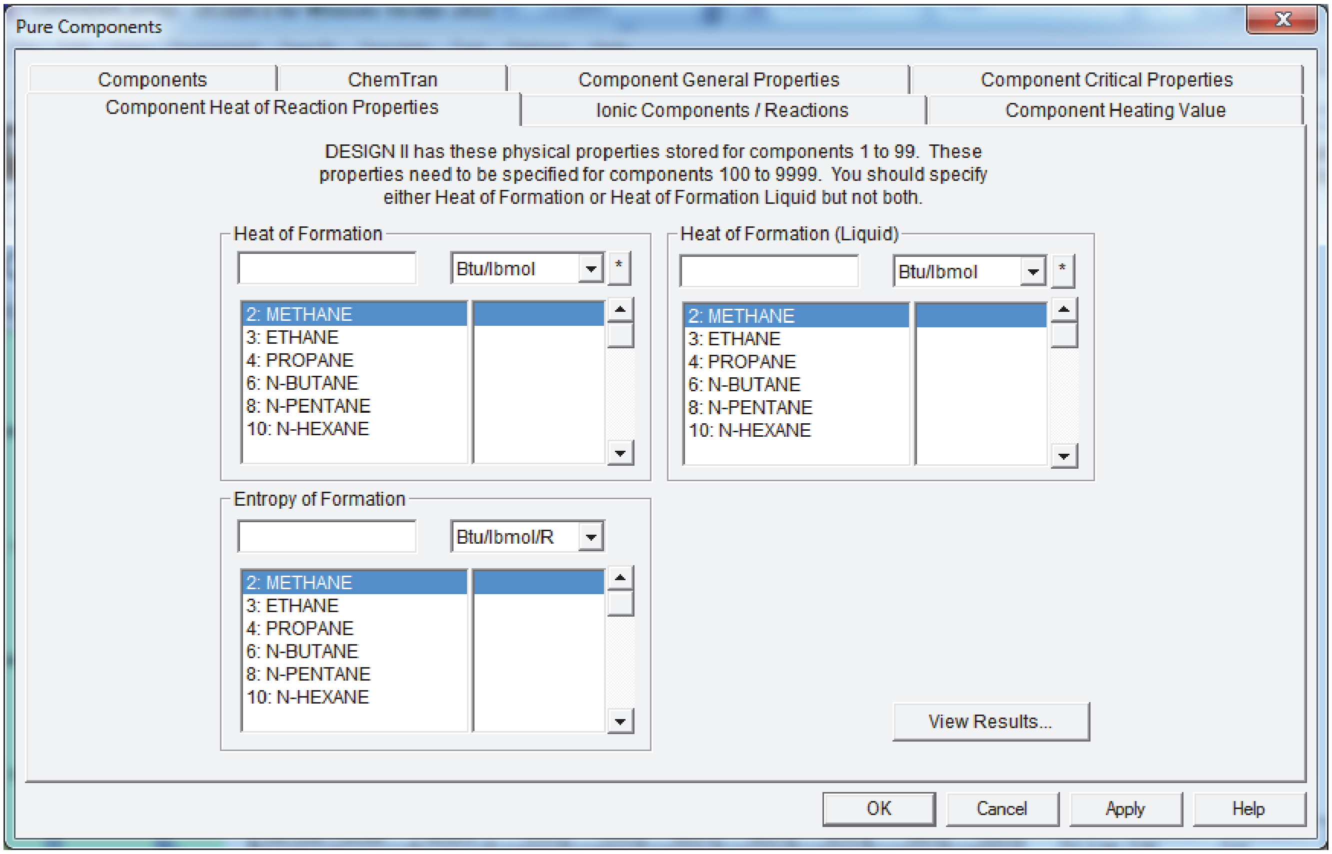


Figure 44: Component Heat of Reaction Properties (from batch2.psd)

Component Heat of Reaction Properties tab: Enter the heat of formation or heat of formation (liquid – if available) and entropy of formation for all components greater than 100 up to 9999.

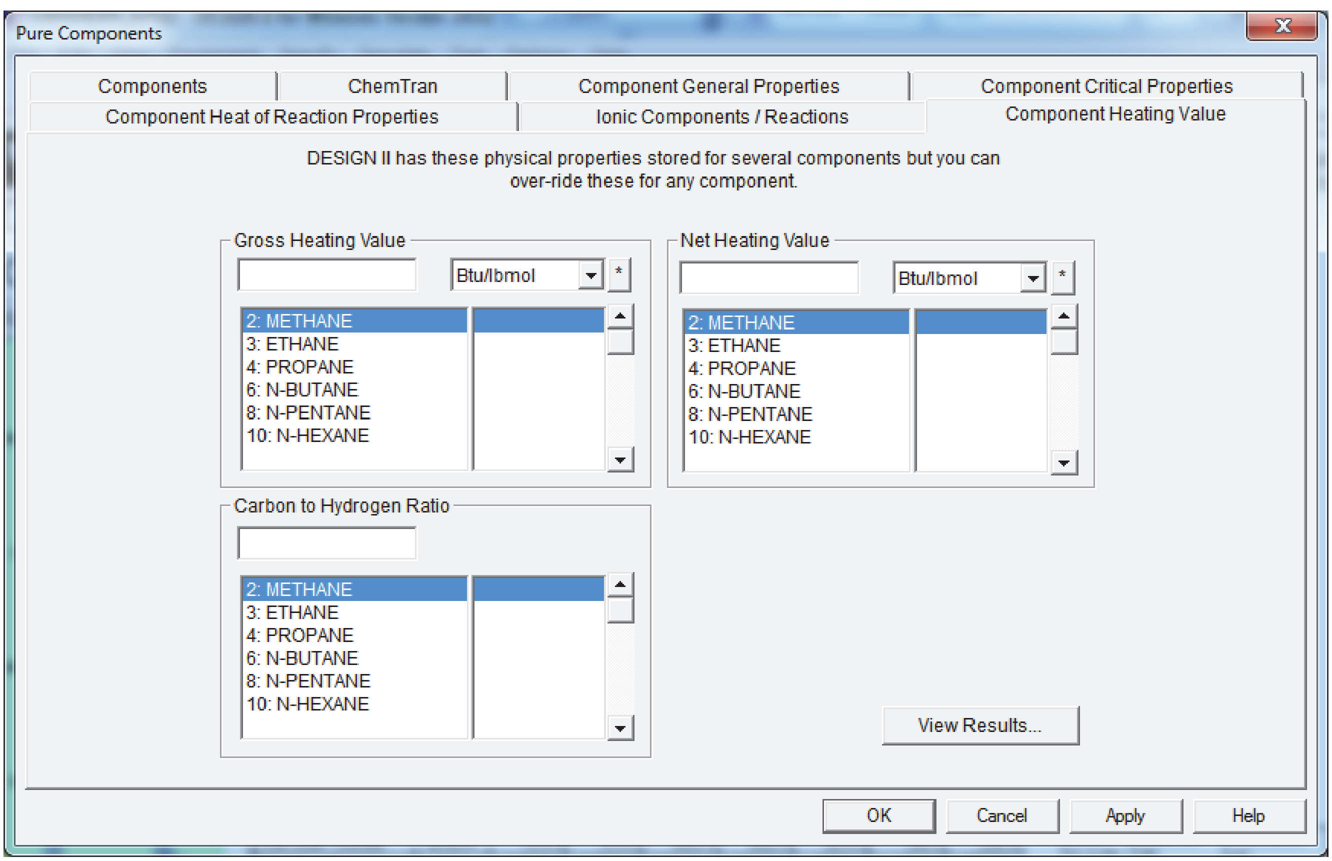


Figure 45: Component Heating Value

Component Heating Value tab: DESIGN II has physical properties stored for several components, but you can override these for any component. Enter the gross heating value, net heating value, and/or the carbon to hydrogen ratio.

Excess Viscosity

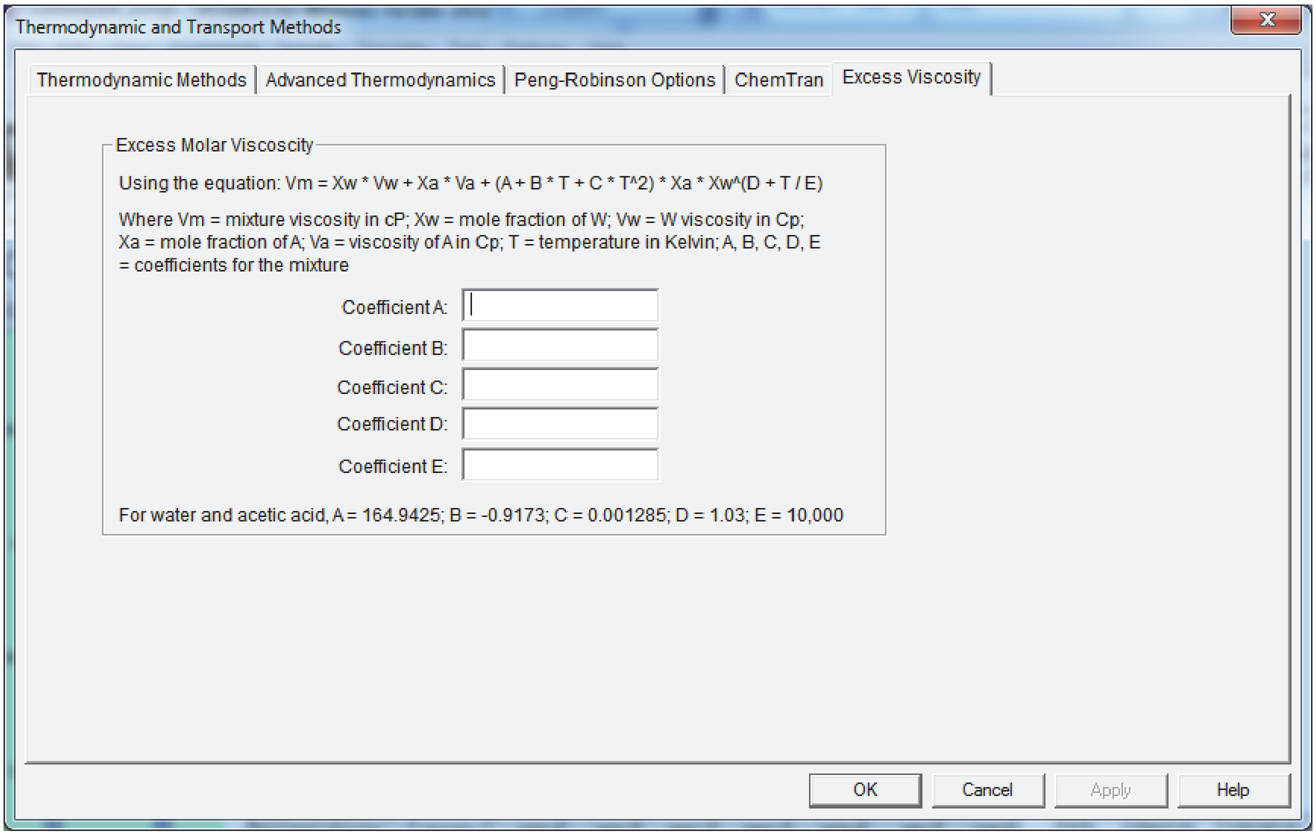


Figure 46: Excess Viscosity tab

Open the **Specify** menu and select **Basic Thermo**. The Thermodynamic and Transport Methods dialog displays with the Thermodynamics tab selected. Select the Excess Viscosity tab.

You can use this tab to enter coefficients in an equation for handling excess viscosity. This uses the equation: Vm = Xw * Vw + Xa * Va + (A + B + C * Tˆ2) * Xa * Xwˆ (D + T/E) where:

Vm = mixture viscosity in cP

Xw = mole fraction of W

Vw = W viscosity in Cp

Xa = mole fraction of A

Va = viscosity of A in Cp

T = Temperature in Kelvin

A, B, C, D, E = coefficients for the mixture

Enter the desired coefficient value. For water and acetic acid, A = 164.9425

B = -0.9173

C = 0.001285

D = 1.03

E = 10,000

Account and Title

Every input file for a flowsheet model must contain an account number and a title. Enter this information using the Specify-Account and Title option.

1. Open the **Specify** menu and select **Account and Title**. The General Data- Account and Title dialog displays.

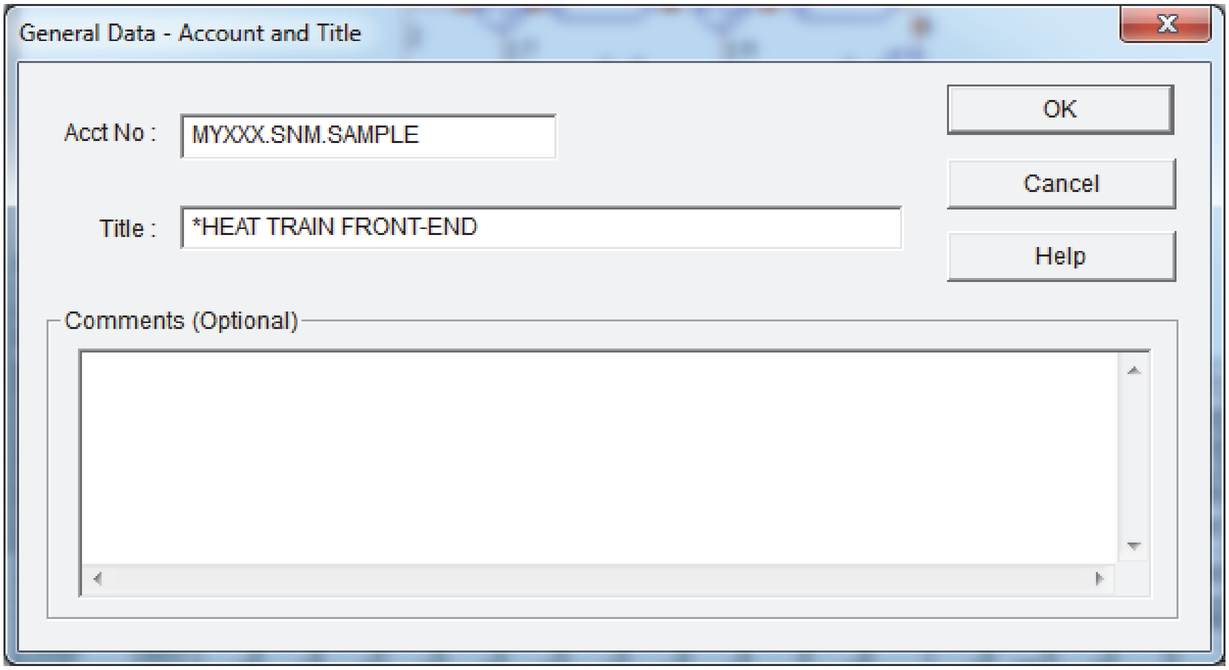


Figure 47: General Data - Account and Title Dialog (from httrain.psd)

2. Enter the following information:

   The account number is defaulted to AB123. If you have been assigned a account number then you may enter your six character Account number followed by a period.

   Enter a title for your flowsheet model. This title is printed at the top of each page of your output file.

   Enter any comments about the flowsheet (optional).

   Click on the **OK** button when done. The Account and Title are set.

Recycle

Use **Recycle** to enter convergence specifications and recycle streams for your flowsheet model. There are several specifications you can use for recycle loop convergence problems allowing you to:

1. Review the sequence of equipment calculations (automatically generated by DESIGN II) and determine the streams for which you will need to provide initial guesses for flowrates, temperature, and pressure.
2. Limit the number of iterations for the recycle loop.
3. Loosen or tighten the tolerance for heat and material balance convergence.

Follow these steps:

1. Open the **Specify** menu and select **Recycle**. The General Data- Recycle dialog displays.

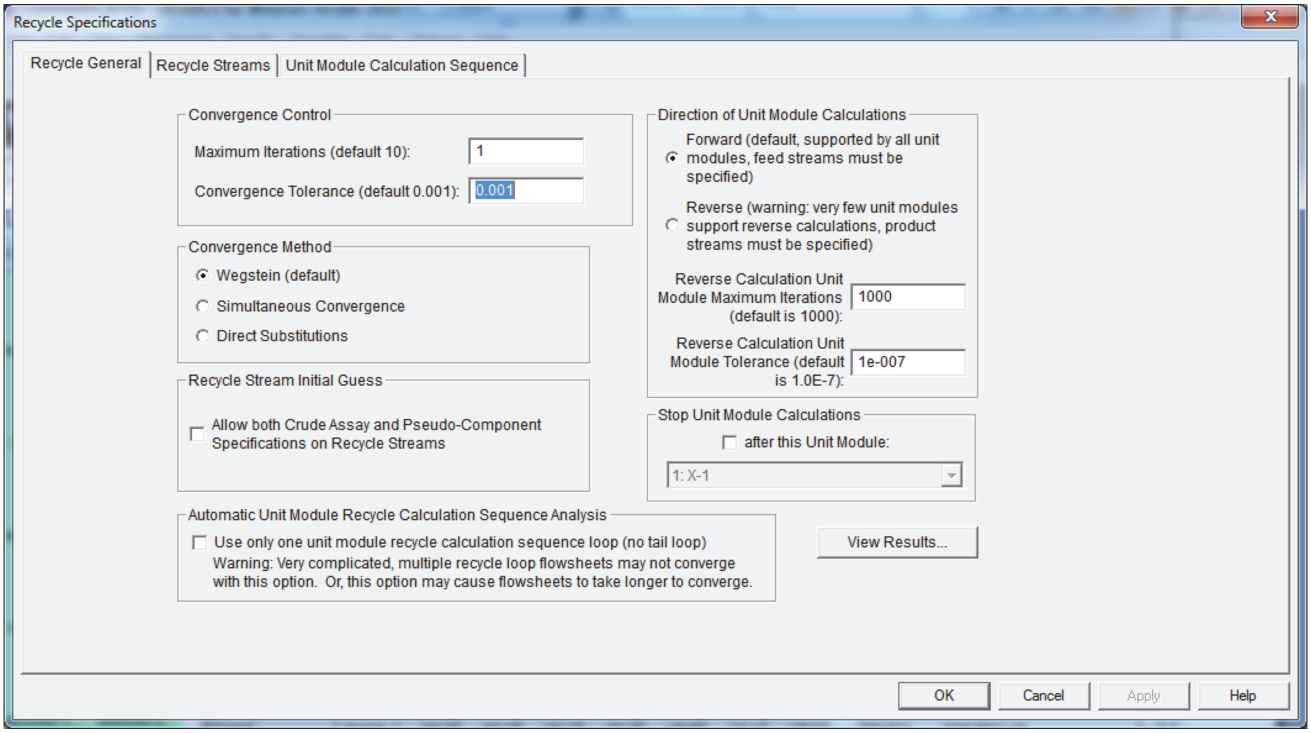


Figure 48: General Data - Recycle Dialog (from expander.psd)

1. Set the desired Recycle options. You can choose **Convergence Control** (options for specifying recycle convergence parameters) or **Convergence Method** (options for the recycle convergence method).

Convergence Control

Maximum Iterations

Enter the maximum number of iterations you want DESIGN II to use to converge the recycle calculations (1 is the default, 10 is recommended).

Convergence Tolerance

Enter the overall heat and material-balance convergence tolerance you want DESIGN II to use (default is .001).

Recycle Streams and Calculation Order

Click this button to set up the streams to recycle and calculation sequence to use. See the next section for details.

Convergence Method

Click on the radio button next to the desired method:

Wegstein

Direct substitution of calculated values will be used for the first two iterations of recycle loop simulation. During the third iteration, DESIGN II will begin using the Wegstein method. This method uses the relevant recycle stream variables from the previous iterations to calculate the variables for the next iteration on a stream-by-stream basis. New values may be dampened, accelerated, or unchanged, based on the analyzed trend for each variable.

Simultaneous Convergence

This method is used primarily for linked columns. This method presumes the recycle streams are closely coupled; recycle stream variables are changed simultaneously to converge recycle loops.

Direct Substitution

This method takes the relevant recycle stream variables calculated from the previous iteration and uses them in the next iteration without adjustment.

Stop Unit Module Calculations

Select this checkbox to stop the unit module calculations after the module you specify (open the drop down list and select a module).

Recycle Stream Initial Guess

Select this checkbox to allow both crude assay and pseudo-component specifications on recycle streams.

Automatic Unit Module Recycle Calculation Sequence Analysis

Select this checkbox to use only one unit module recycle calculation sequence loop (i.e. no tail loop). See the warning on the dialog for more details.

2. Click **OK** when done. The General Data- Recycle dialog closes.

Recycle Streams and Calculation Order

If you click on the **Recycle Streams and Calculation Order** button on the **General Data – Recycle** dialog, the **General Data – Recycle Streams and Calculation Sequenece** dialog appears. The streams you select are the minimum recycle streams for which component flowrates are to be accelerated using either Wegstein or the Simultaneous Convergence techniques (see ***Convergence Method***). These are the streams for which you will enter guesses for component flowrates, temperatures, and pressures using the Stream-Basics dialog.

1. Select the **Specify the Recycle Streams** option.
2. The first list box in this dialog contains a list of all the streams in your flowsheet. The second box lists the streams you have selected for your recycle streams. You put a stream into the second list box by selecting it from the first list box. There are several ways to select streams for the model:

   Select streams using the mouse (in the top list box, double-click on the stream name, or click on a stream name and click the **Add** button). To remove a stream from the list of recycle streams, click on the stream name in the bottom list box; then click the **Delete** button. You can scroll either the top or bottom list, if necessary. To change the stream order in the bottom list box, first click on the stream; then click the **To Top** or **To Bottom** button.

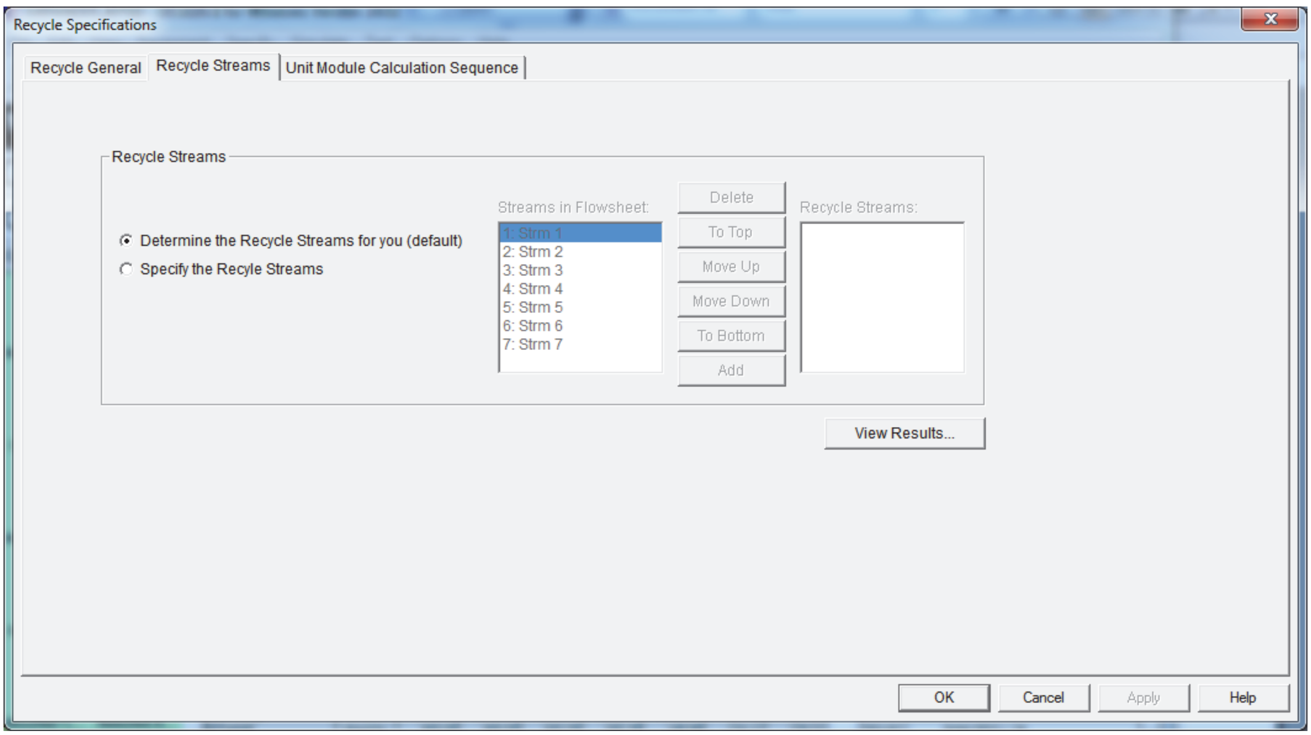


Figure 49: General Data - Recycle Streams Dialog (from expander.psd)

Unit Module Calculation Sequence

Select one of the following:

Let DESIGN II determine the Calculation Sequence and All Recycle Loops for you: leave this set to have the program set the sequence and recycle loops.

Specify a Calculation Sequence: complete the Calculation Sequence Section.

Specify a first Recycle Sequence; then an Optional Second Recycle Sequence: complete the Recycle Sequence 1 and Recycle Sequence 2 sections.

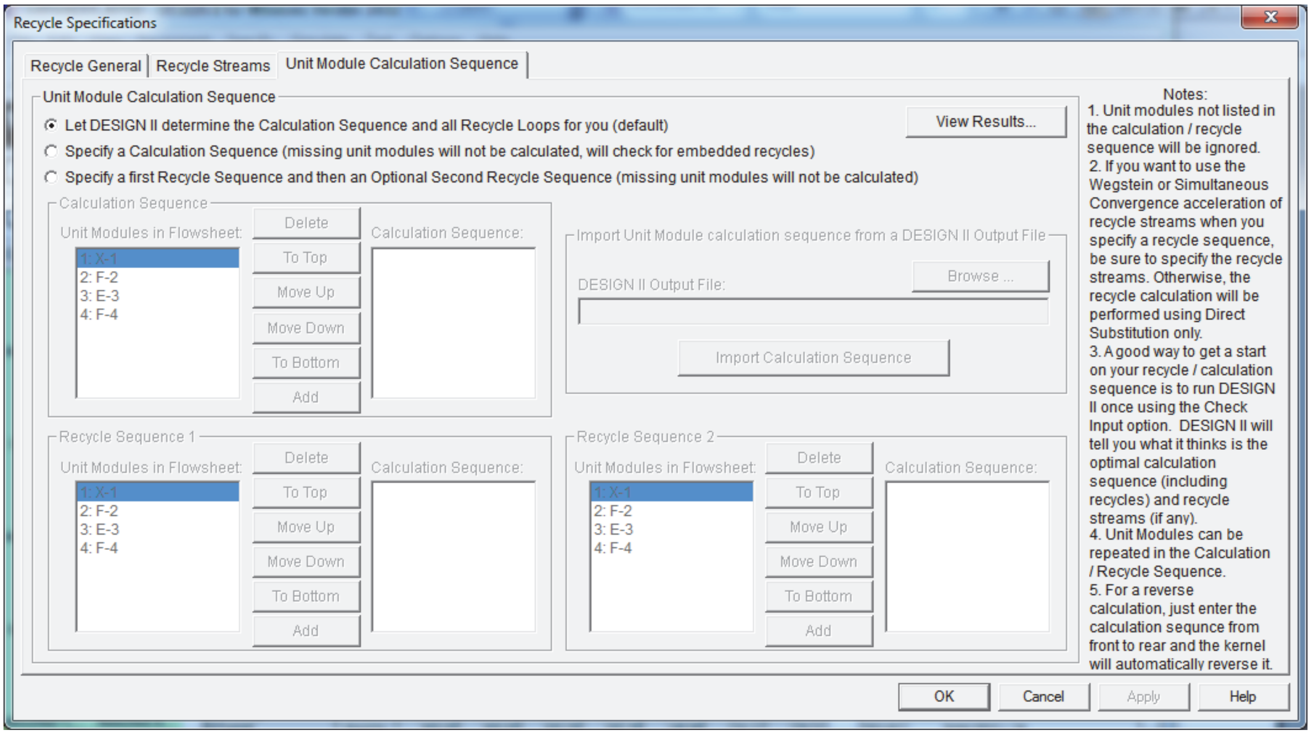


Figure 50: Unit Module Calculation - Recycle Streams Dialog (from expander.psd)

Calculation Sequence

Click on a unit name from the Unit Modules in Flowsheet list and click the Add button to move it to the Calculation Sequence list (you can also double click on it). To remove a unit from the Recycle Streams list, click on its name then click the Delete button. To change the order of a unit in the calculation sequence, click on its name in the Calculation Sequence list then click either To Top, Move Up, Move Down, or To Bottom.

Recycle Sequence 1 and 2

Click on a unit name from the Unit Modules in Flowsheet list and click the Add button to move it to the Calculation Sequence list (you can also double click on it). To remove a unit from the Recycle Streams list, click on its name then click the Delete button. To change the order of a unit in the calculation sequence, click on its name in the Calculation Sequence list then click either To Top, Move Up, Move Down, or To Bottom.

Import Unit Module calculation sequence from a DESIGN II Output File

You can choose to import an existing unit module calculation sequence from a DESIGN II output file. Use Browse to locate the desired file to import (with an extension of .out), click on its name, and click Open. The name of the file appears in the field. Click the Import Calculation Sequence button to import the calculation sequence from the file. Please view the notes on the dialog for more details.

3. Click on the **OK** button when done to return to the General Data- Recycle dialog.

Print Options

You may set two print options: Reports Sections and Units System.

The Reports Sections option controls whether the Stream Summary and Detailed Stream Print sections are added to your flowsheet printout, and/or a CSV (Comma Separated Value) file is created during program execution.

The Units System option allows you to select the type of units used for the flowsheet printout: US, SI, or Metric. You can also override specific units for Temperature, Pressure, Enthalpy, Time, etc. (mixing unit systems).

To set the print options:

1. Open the **Specify** menu and select **Print Options**. The General Data- Print Options dialog displays.

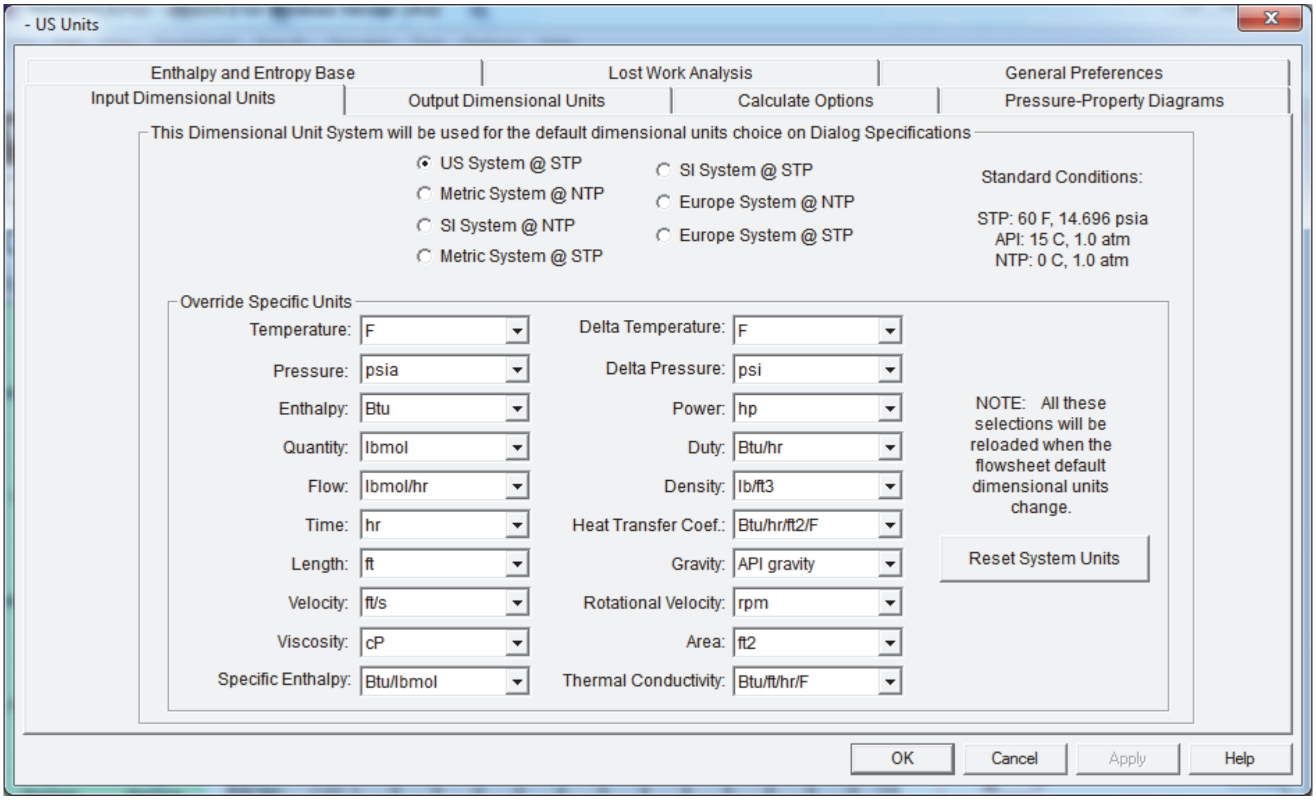


Figure 51: General Data - Print Options Dialog (from refinery.psd)

Click on the **check box** next to the desired Reports Sections you want (you can select from none to all three), and the **radio button** next to the Units System you want (select only one). To set your own units, click on the **Override Specific Units** button. The General Data Output Units dialog displays.

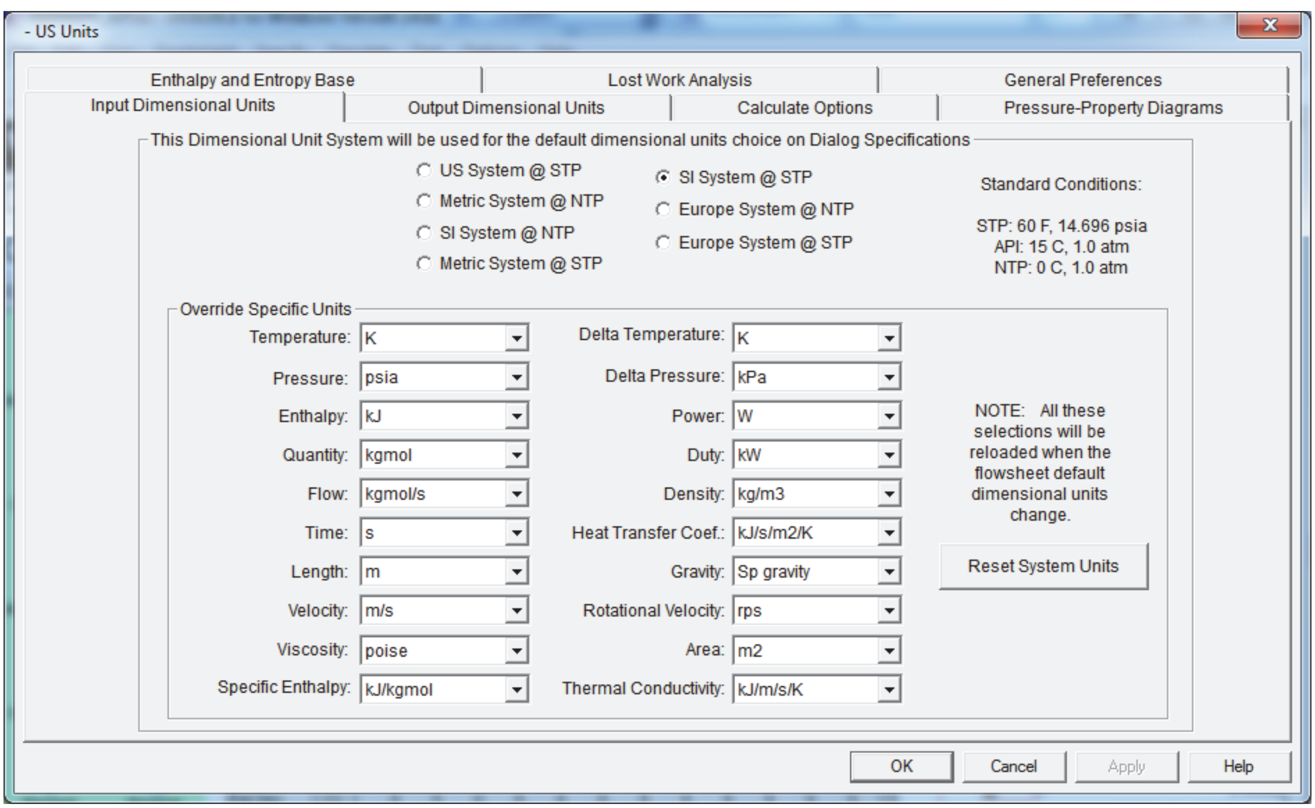


Figure 52: General Data - Output Units Dialog (from expander.psd)

Click the **check box** next to the label for which you want to set a specific unit; then open the drop down list next to the item and select the desired unit. When done, click the **OK** button to save your changes to specific unit options.

Dynamic Settings

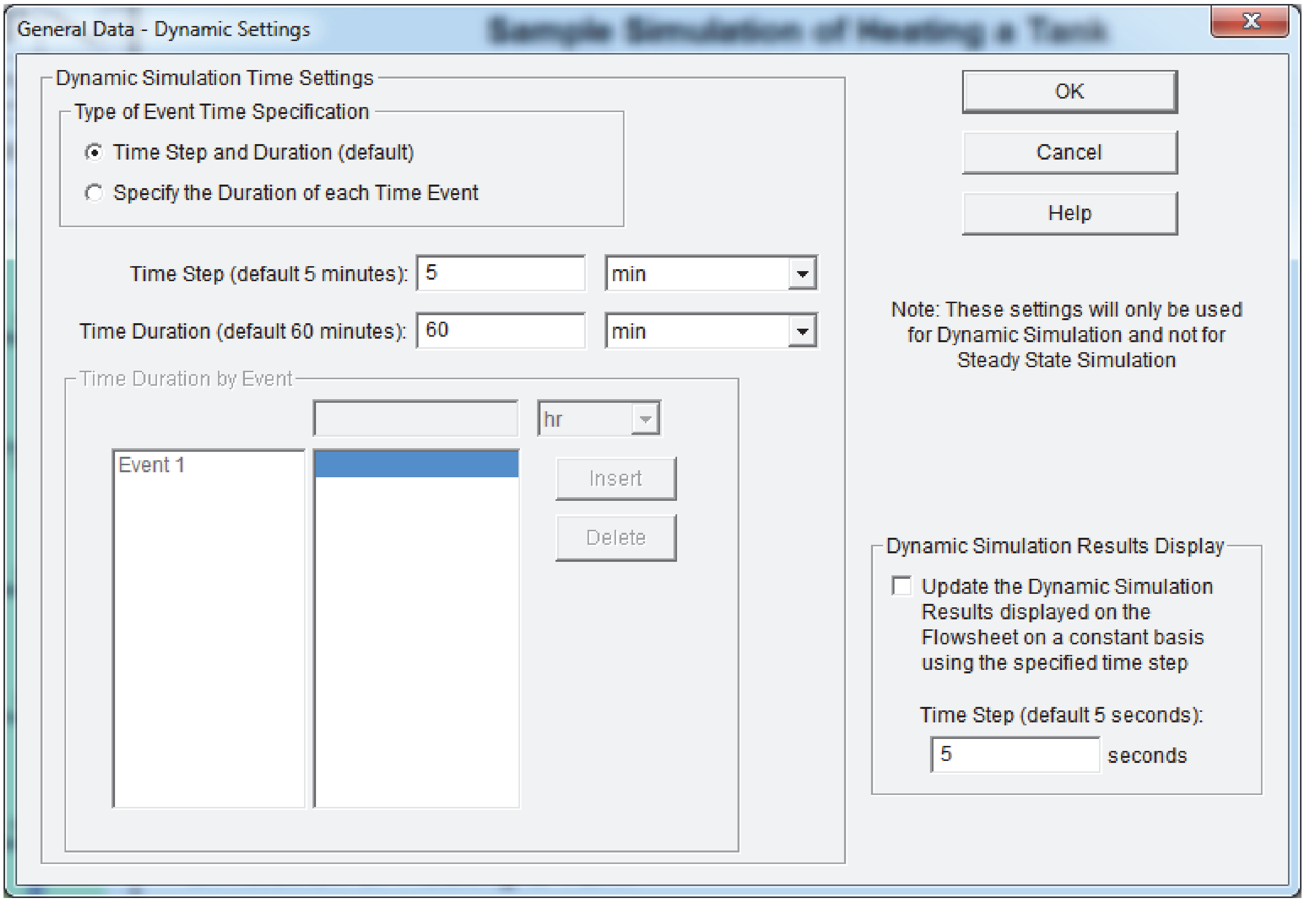


Figure 53: General Data – Dynamic Settings Dialog (from tank1.psd)

Open the **Specify** menu and select Dynamic Settings. The General Data- Dynamic Settings dialog displays.

Use these settings when running a dynamic simulation; these are not used for Steady State simulations. Enter a time step and duration. See Chapter 10 Simulation for more details.

Optimization

Use Optimization to enter commands that cause DESIGN II to automatically search for the optimal values of the selected parameters, to maximize or minimize the optimization function.

This section describes the optimization steps involved in systematically solving an optimization problem.

1. Base case Simulation - OPTIM first runs the base case flowsheet simulation using the values of design variables from the previous iteration (or initial guesses supplied by the user if it is the first iteration). When the base case simulation is finished the objective and constraint functions are calculated and stored.

2. Perturbation - OPTIM adjust (perturbs) the value of one variable by an amount which is controlled by the PER ABS or PER REL command. Then the simulation is rerun using the new value and new objective and constraint functions are calculated. If there are multiple variables, this step is repeated for each variable with all other variables at their base case values. The values of these perturbations are included in the output file.

3. Search Vector - OPTIM analyzes the amount of change in the objective and constraint functions with respect to each variable perturbation and determines the search vector-new values for each variable to be used in the evolutionary simulation. The new values are reported in the output file.

4. Evolutionary Simulation - OPTIM now performs the simulation using the new design variable values and calculates the new objective and constraint functions. OPTIM will then either start another iteration until a solution is obtained, based on the solution tolerance (discussed later), or generate a message explaining why an optimal solution was not obtained.

NOTE: The solution of an optimization problem often depends on the strategic use of scaling, perturbation sizes, and maximum search steps. A few case studies can provide information that leads to a choice of the variables which will reduce the overall computation time.

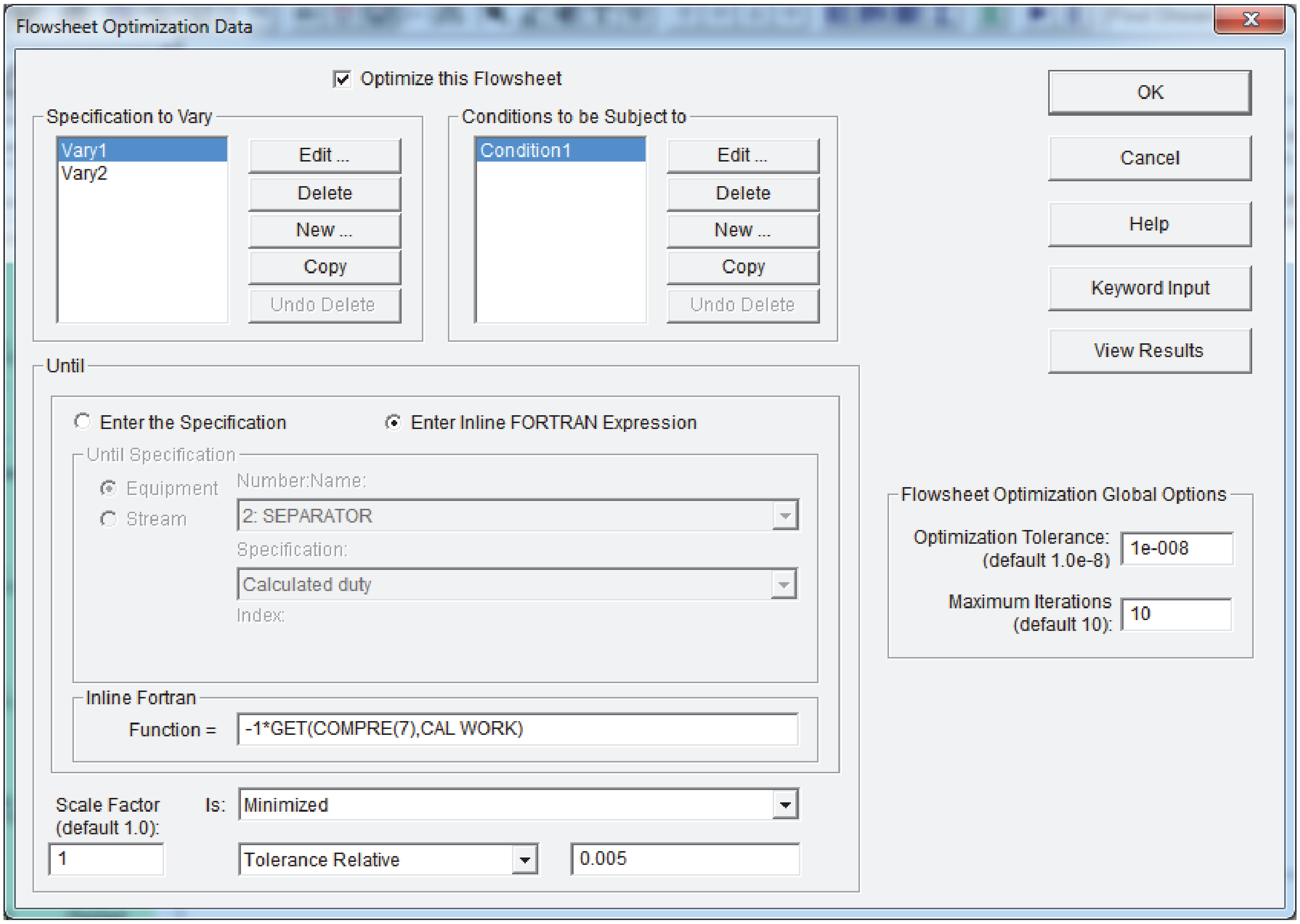


Figure 54: Flowsheet Optimization Data Dialog (from optimiz1.psd)

Open the **Specify** menu and select **Optimization**. The Flowsheet Optimization dialog displays.

Every optimization problem is required to have an objective function. The objective function is a function of equipment parameters or stream properties that is to be made as small (MINimized) or as large (MAXimized) as possible.

The solution tolerance will be the default tolerance unless it is changed by using one or more of the tolerance commands. The optimization calculation continues to iterate until it attains an optimal solution as defined by any one of the following occurrences:

1. A consecutive iteration fails to improve the objective function by more than a given tolerance, on condition that all the constraints (including variable bounds) are reasonably met. Objective function tolerance is specified using the TOLerance ABSolute or TOLerance RELative command which follows the keyword FUN (or its explicit definition). If only TOLerance is entered, it is assumed to be TOLerance ABSolute. For TOLerance RELative, the tolerance is applied to the prevailing value of the objective function. The default is a relative tolerance of 0.005.

2. The change in every design variable for consecutive iterations is less than a given tolerance. Design variable tolerances are specified using the TOLerance ABSolute or TOLerance RELative command following the definition of variables. If only TOLerance is entered, it is assumed to be TOLerance ABSolute. Tolerances may be specified as ABSolute values for some variables, and as RELative values for others. For the RELative option, the tolerance applies to the specified range of the design variable, which is defined as the difference of the corresponding upper bound and lower bound. If either bound is missing, the tolerance applies to the initial guess for that variable. The default is a relative tolerance of 0.001 for every variable.

3. An implicitly defined non-negative function becomes smaller than the OPTimization TOLerance. This function is derived from the Karush-Kuhn-Tucker conditions. The default value for OPTimization TOLerance is 10-8. Normally, this command will not be needed.

DESIGN II execution will also terminate when better designs cannot be located along the search vector, even if none of the above criteria are satisfied. In this case, a decision must be made about whether the solution at the last iteration is satisfactory. If it is not, another set of initial guesses and/or algorithmic parameters must be tried.

Proper scaling is important in achieving good optimization results. Design variables, constraints and the objective function each have a SCAle command. A MULtiply command is also available for constraints. The effect of scaling is different for each.

Scaling of design variables changes neither the problem nor the true solution. It does, however, alter the topology of the problem. Essentially, larger scaling factors yield steeper gradients of objective function with respect to the corresponding variables. The default SCAle factor for design variables is 1.0.

The objective function can also be scaled. Here, the objective function is actually changed but the true solution of the design variables remains the same if the scaling factor is positive. A higher objective scaling factor gives a steeper response surface in all directions.

Constraints can be scaled by factors you enter with MULtiply commands. Both sides of the constraint expression will be multiplied by these factors Constraint multiplication does not change the true solution. It is used to emphasize or de-emphasize individual constraints, especially when equality (.EQ.) constraints are involved. Note the difference between MUL factors and SCAle factors for constraints: the SCAle factor only applies to the value entered as a constant and will very likely to change the true optimum. The MULtiply command can be used to ensure that the constraint value(s) will be the same order of magnitude as the objective function value.

NOTE: Only the objective function can have a negative scale factor.

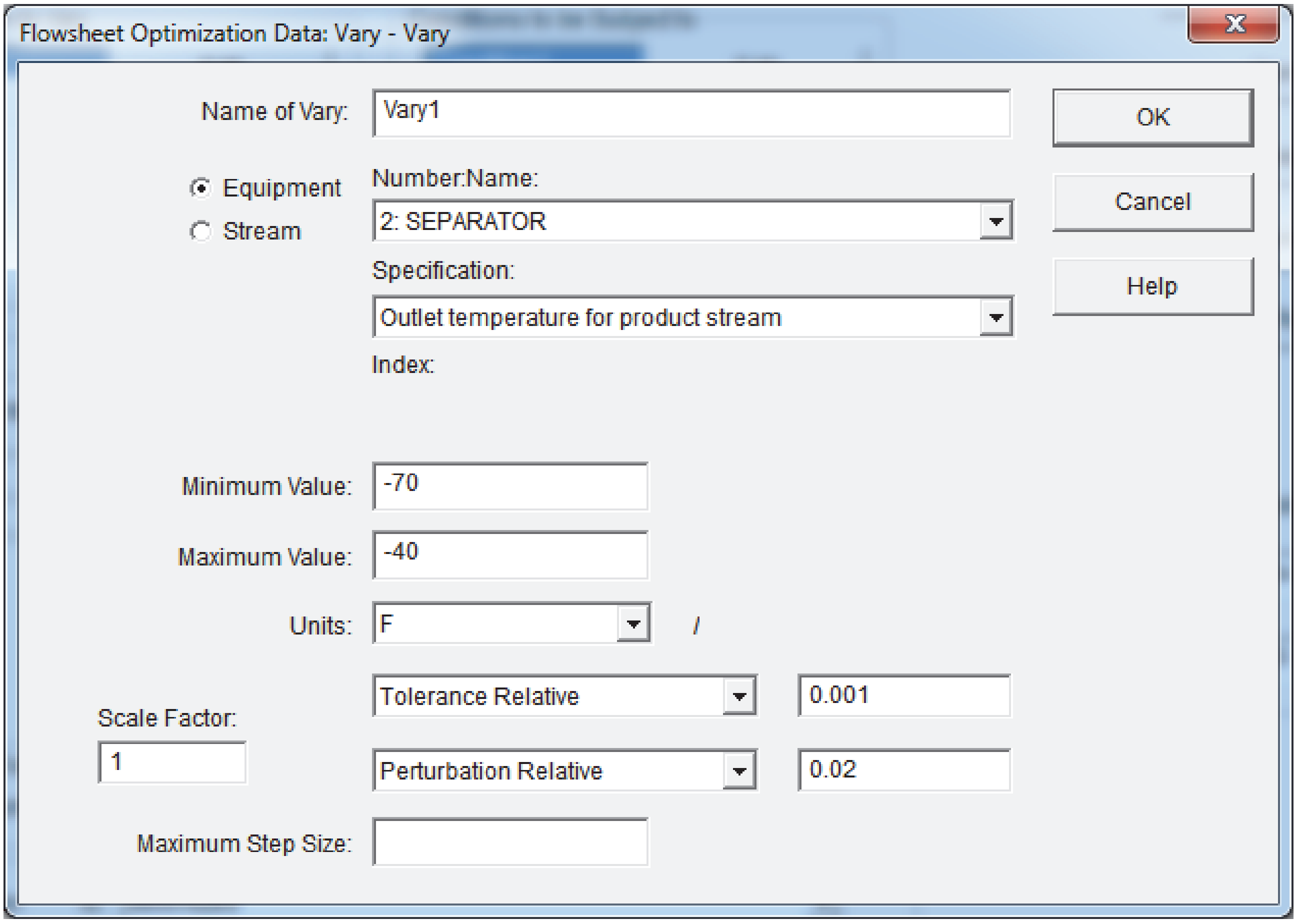


Figure 55: Flowsheet Optimization Data - Vary Dialog (from optimiz1.psd)

Design variables are defined as the stream properties or equipment parameters whose values are to be determined in order to optimize the objective function. The first design variable is preceded by the keyword VARy. Not every stream property or equipment parameter qualifies as a design variable. In the first place, it should be an item whose value can be specified in the input section (i.e. flows for specific components, total stream flow, outlet temperature for a heat exchanger, etc.). The CALculated WORK for an expander cannot be entered in the input file; consequently, it cannot be used as a design variable. A complete list of design variables for equipment modules and streams is shown in the DESIGN II General Reference Guide.

At least one design variable is required in every optimization problem. There is no limit to the number of design variables; however, computation time increases rapidly with an increase in the number of design variables.

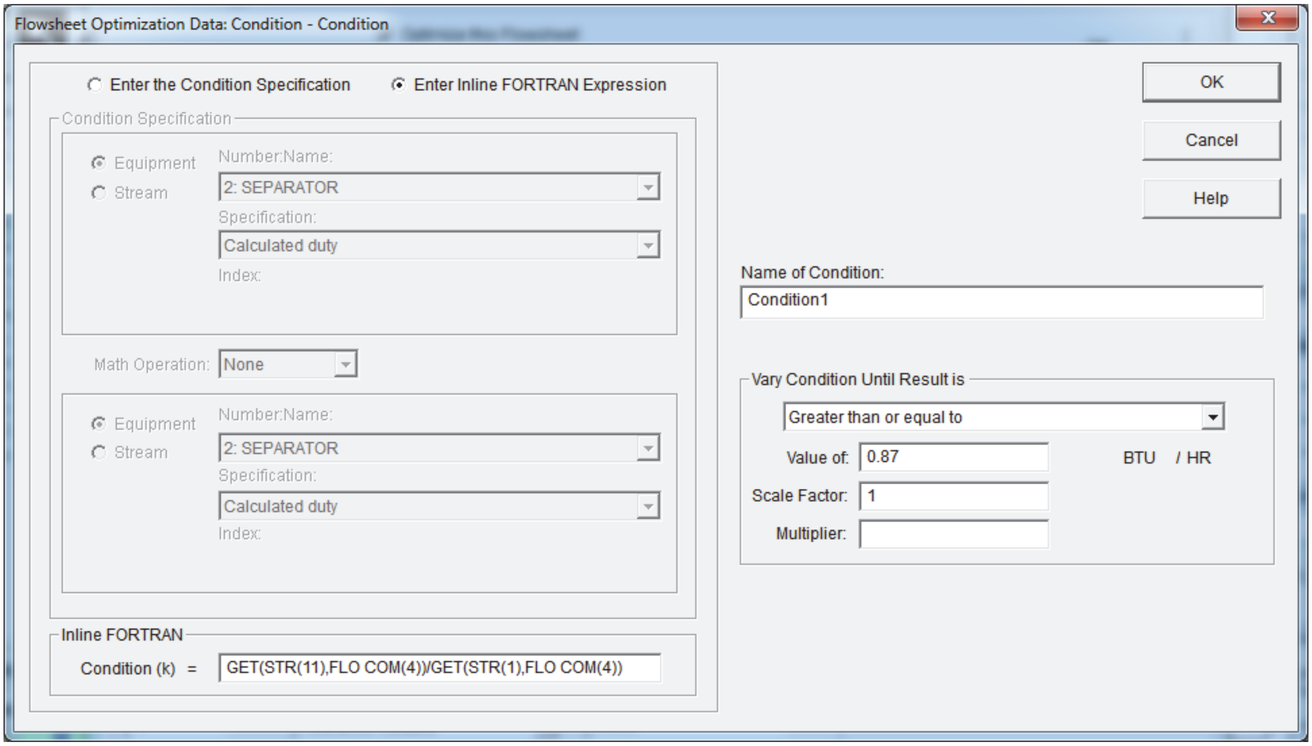


Figure 56: Flowsheet Optimization Data - Condition Dialog (from optimiz1.psd)

Strictly speaking, simple bounds to design variables are also constraints to optimization

Both constraints and objective function can be entered as Inline FORTRAN statements beginning with F- or they can be defined with Inline FORTRAN expressions explicitly. Both implicit and explicit statements can be used in the same run. Each CONstraint which is entered using the keyword CON (i.e. not defined explicitly) must be numbered between 1 and 100. There is no requirement that the numbers be in ascending or sequential order. The maximum number of constraints (implicit and explicitly defined) is 100. The number of independent equality constraints cannot exceed the number of design variables.

Constraints are evaluated using the DESIGN II internal units set. The numerical value for the constant should also be in internal units. Briefly, the internal units set is

Temperature: degrees R
Pressure: PSIA
Quantity: lbmoles
Heat, energy: BTU
Length: feet
Time: hours

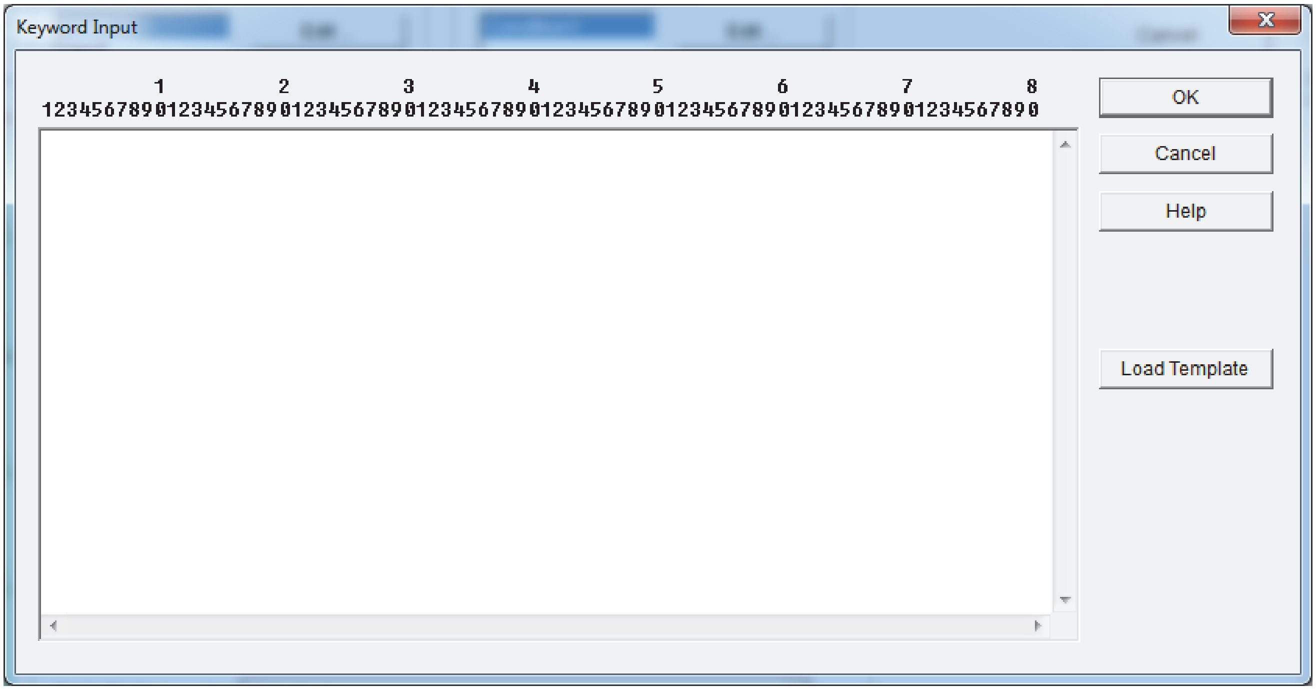


Figure 57: Flowsheet Optimization Data - Keyword Input Dialog (from optimiz1.psd)

1. Open the **Specify** menu and select **Optimization**. The Flowsheet Optimization Data dialog displays. Select the Keyword Input button.
2. Enter the desired keyword input, or click the **Load Template** button to load a pre-defined template of keywords. This dialog is similar to the one used for setting Optional General specs: Keyword Input. Refer to the ***Setting Optional General specs: Keyword Input***, ***Creating Keyword Commands***, and ***Editing Keyword Commands*** sections for details.
3. Click the **OK** button when done. The Flowsheet Optimization Data- Keyword Input dialog closes.

Size Lines

Use Size Lines to allow DESIGN II for Windows, using client-defined criteria, to calculate pipe sizes for streams; three diameters will be reported: target nominal diameter and next larger and smaller pipe diameters.

Please note that each stream supports its own size line dialog. This dialog is maintained as an obsolete feature.

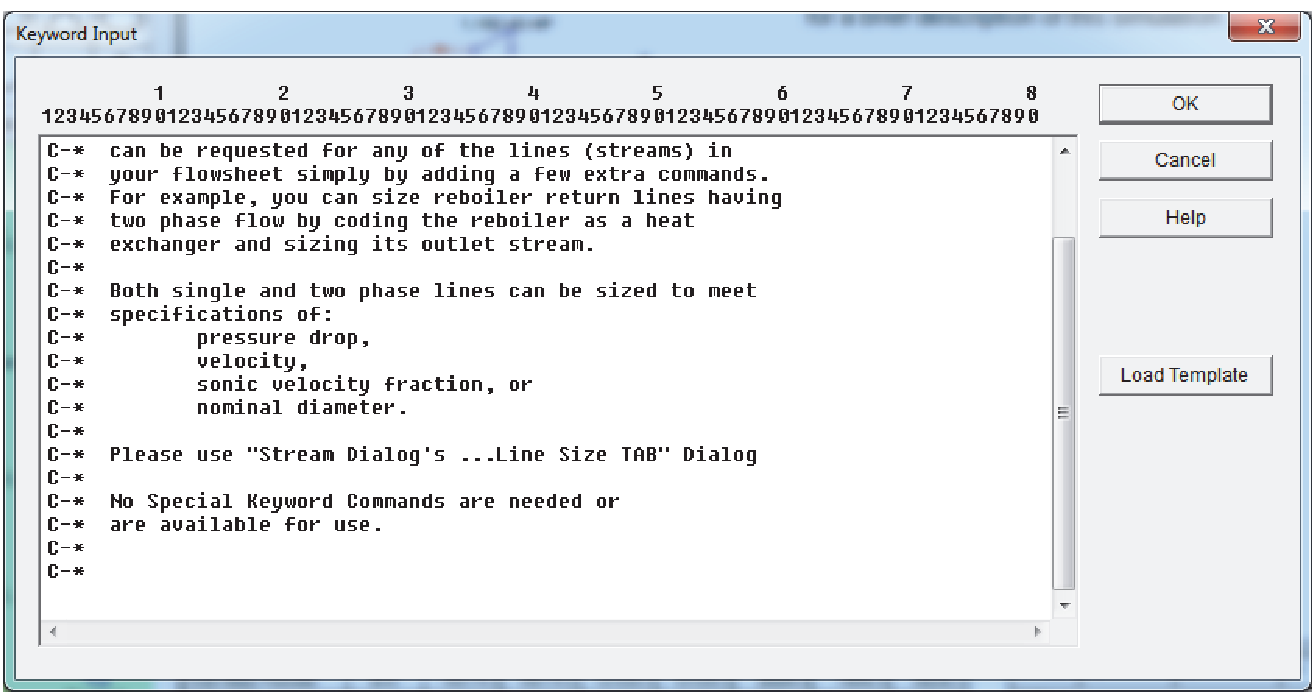


Figure 58: Size Lines Data - Keyword Input Dialog (from expander.psd)

1. Open the **Specify** menu and select **Size Lines**. The Size Lines Data- Keyword Input dialog displays.
2. Enter the desired keyword input, or click the **Load Template** button to load a pre-defined template of keywords. This dialog is similar to the one used for setting Optional General specs: Keyword Input. Refer to the ***Setting Optional General specs: Keyword Input***, ***Creating Keyword Commands***, and ***Editing Keyword Commands*** sections for details.
3. Click the **OK** button when done. The Size Lines Data- Keyword Input dialog closes.

Inline Fortran

Use this option to enter Inline Fortran instructions for the Fortran library section of the DESIGN II input. Each section of the Inline Fortran template contains general instructions. For additional instructions about using Inline Fortran, please refer to the online ***DESIGN II User Guide***.

Open the **Specify** menu and select **Inline Fortran**. The General Data- Inline Fortran dialog displays.

You may begin typing Inline Fortran statements directly in the edit box, or you may follow the remaining steps below.

NOTE: If you type commands directly in the edit box, begin each Fortran line with 6 blank spaces. This is handled automatically in the Fortran template.

1. Open the drop down list next to **Select Inline Fortran Template Type** and select the desired template type.
2. Enter the desired keyword input, or click the **Load Template** button to load a pre-defined template of keywords. Frequently, Inline Fortran statements include parameters from more than one equipment type, or from streams, or use predefined subroutines. You may load more than one template type in the edit box. To make your Fortran statements easier to read, delete unused commands before loading the next Fortran template type.
   This dialog is similar to the one used for setting Optional General specs: Keyword Input. Refer to the ***Setting Optional General specs: Keyword Input***, ***Creating Keyword Commands***, and ***Editing Keyword Commands*** sections for details
3. Click the OK button when done. The General Data- Inline Fortran dialog closes.

Case Study

Use Case Study to allow DESIGN II for Windows, using client defined criteria, to calculate numerous flowsheet runs while varying one or more flowsheet variables.

The CASe Study section allows an engineer to run a number of similar simulation cases at once in order to assess the sensitivity of a flowsheet to changes in variables or to produce a number of design cases. CASe study allows you to present your results in data tables and plots for efficient analysis. This has the effect of reducing both the engineering and computer time required.

There is a limit of 25 cases that may be specified in one CASe study and only one CASe study section per flowsheet. All parameter changes will be implemented and the entire flowsheet is recalculated with the new parameter setting.

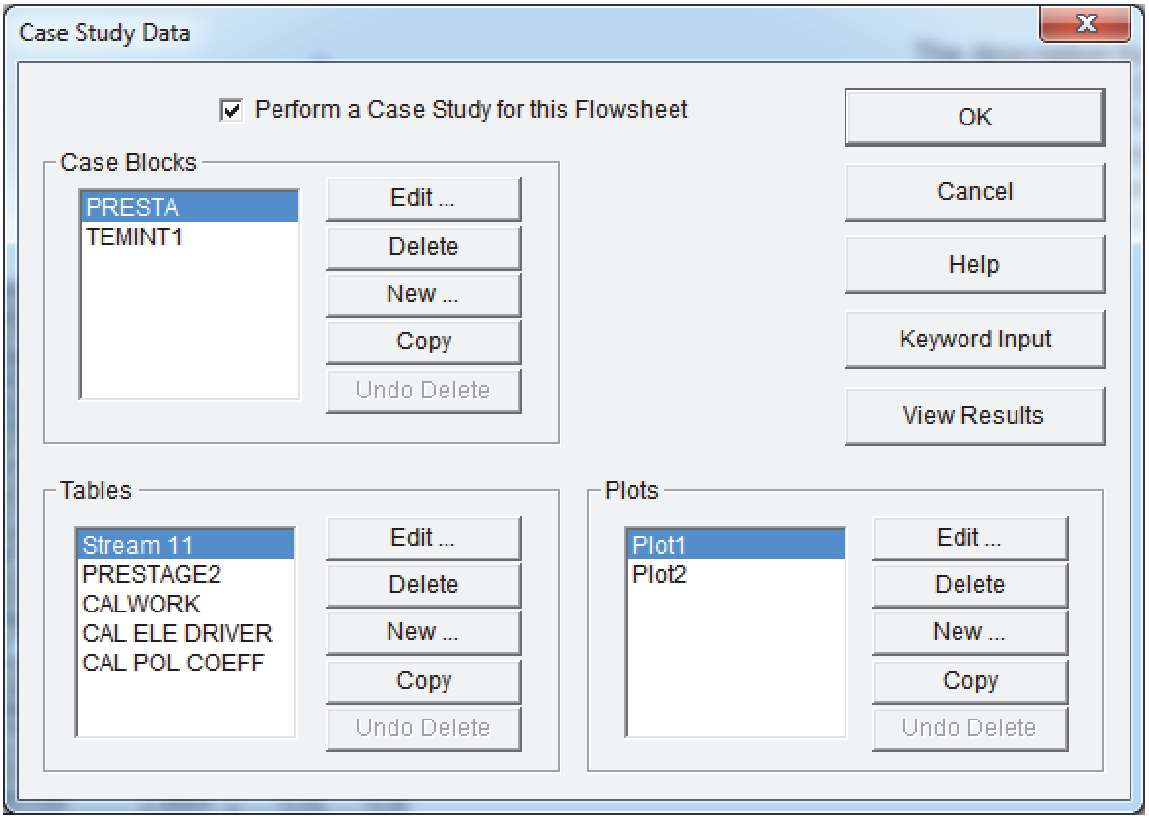


Figure 59: Case Study Data Dialog (from case1.psd)

CASe studies, by their nature, tend to produce extremely large output files. Unless the detailed information is required, use the GENeral section, PRInt SWItch = 3 and PRInt STReam - NONE options to reduce the volume of output.

There is an overall limit of 25 cases that may be specified in one run. There will normally be one case per CASe block; however, if a STEp command is used, a CASe is run for each increment. A maximum of 10 cases may be generated by one STEp command. This is to prevent a poor specification from effectively causing an infinite log.

CASe study uses the converged solution from one calculation as the starting point for the next. Because of this, it is advantageous to arrange the case studies in such a way that the step (or change) from one case to the next is as small as possible. This will both reduce the amount of computer time required and increase the likelihood of convergence

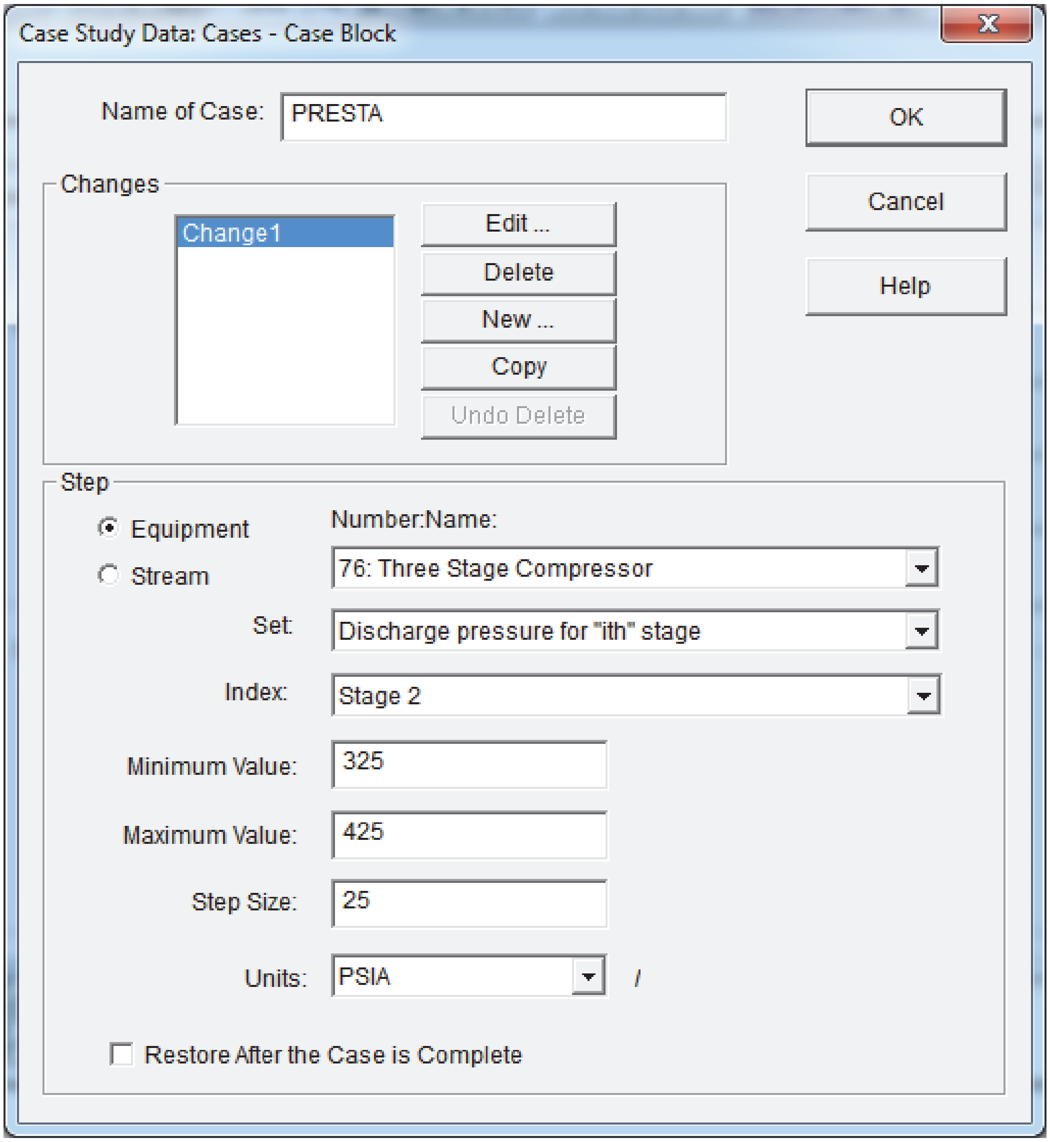


Figure 60: Case Study Data – Case Block Dialog (from case1.psd)

The CASe block command marks the beginning of a new input case. Each CASe block may specify any number of flowsheet parameter changes. All parameter changes will be implemented and the entire flowsheet is recalculated with the new parameter setting. There is a limit of 25 cases that may be specified in one CASe study. NAMe = up to 16 characters.

The CHAnge command is used to modify the value of a parameter in the flowsheet. A list of valid parameters is given in Tables 2 and 3 of the Inline FORTRAN section.

The CHAnge command may optionally be followed by the REStore keyword. This has the effect of restoring the parameter value to its original value after the current case is completed.

The STEp command is a convenient way to specify a series of cases where a parameter is being varied in a systematic fashion. This generates a set of cases, with the parameter being incremented by the amount specified for each successive case until the specified limit is exceeded. This is equivalent to a series of CHAnge commands. A list of valid parameters is given in Tables 2 and 3 of the Inline Fortran section.

The STEp command may optionally be followed by the REStore keyword. This has the effect of restoring the parameter value to its original value after the current case is completed. If the increment is negative, the limit becomes a lower limit. There is a limit of 10 cases that may be run with a single STEp command.

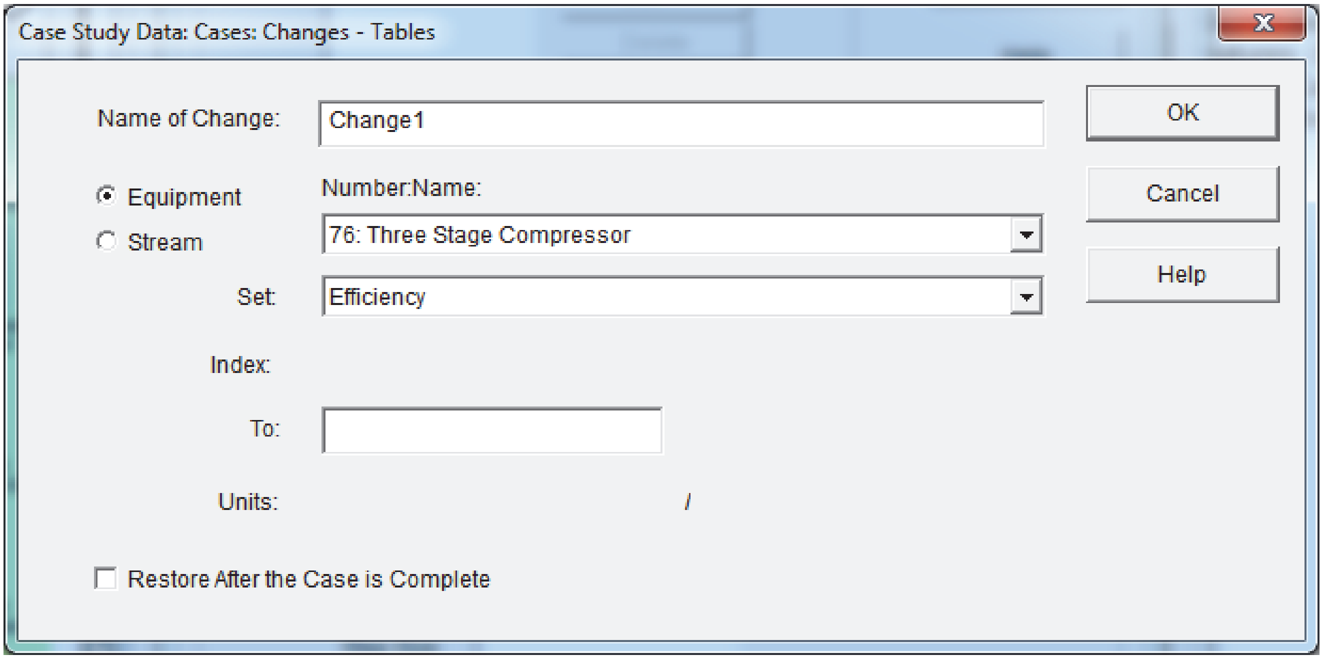


Figure 61: Case Study Data – Changes - Tables Dialog (from case1.psd)

The CHAnge command is used to modify the value of a parameter in the flowsheet. A list of valid parameters is given in Tables 2 and 3 of the Inline FORTRAN section.

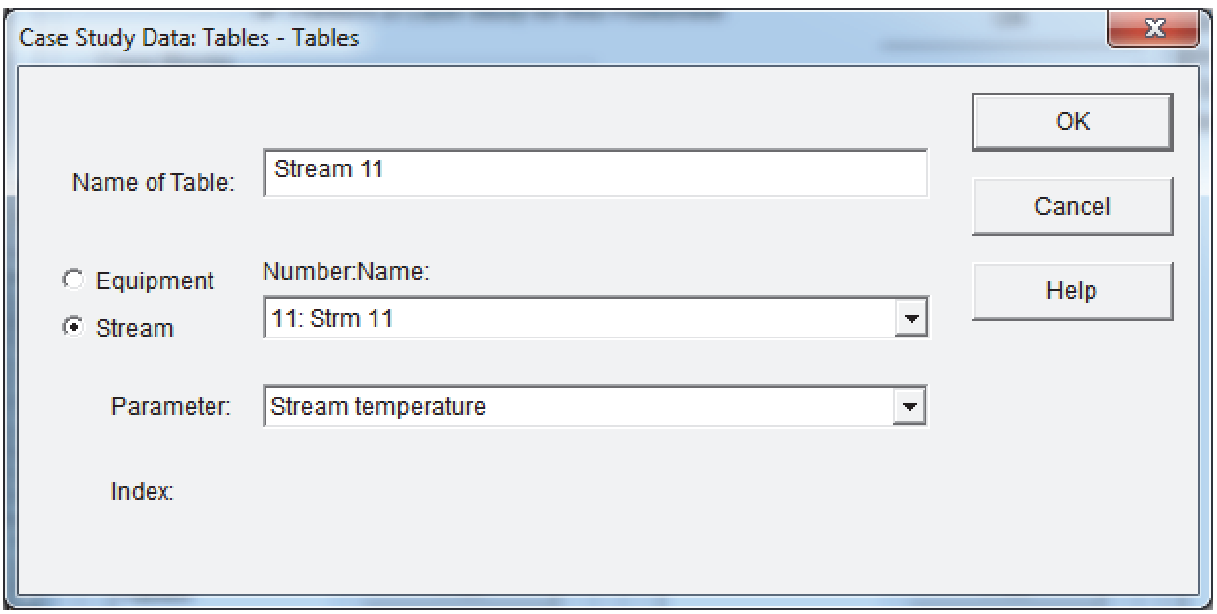


Figure 62: Case Study Data - Tables Dialog (from case1.psd)

The TABle command allows you to build a comparative table for case studies. All variables which occur in CHAnge or STEp statements will automatically be reported in the table. The user may add to this list by using the TABle command. The list of valid parameters is identical to the parameters available to Inline FORTRAN and is called using the same keywords that are used by Inline Fortran (Tables 2 and 3).

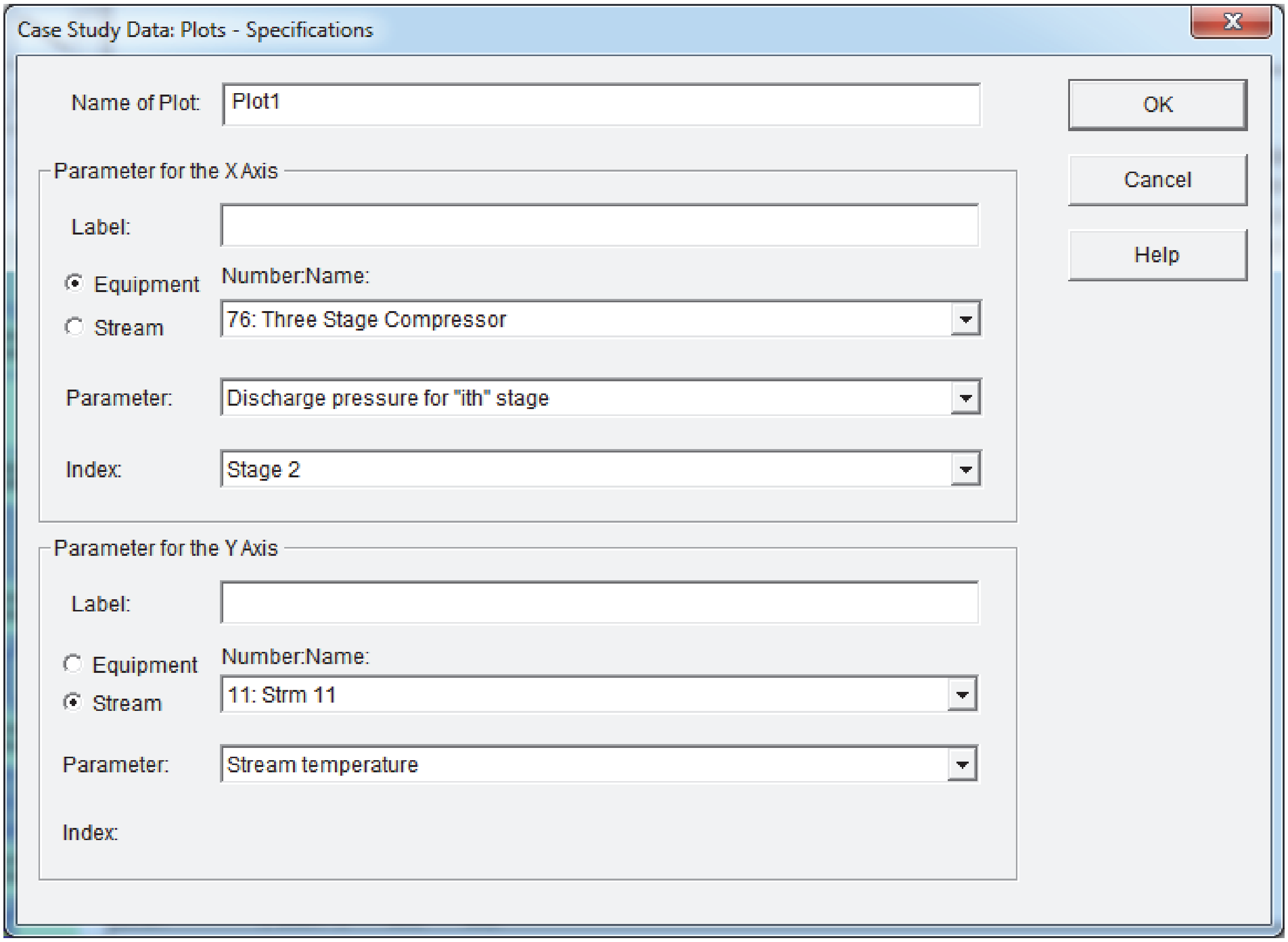


Figure 63: Case Study Data – Plot Dialog (from case1.psd)

The PLOTX/PLOTY commands allow you to graph the changes in one variable against another. The PLOTX command defines the independent variable and the PLOTY defines the dependent variable. UP to four dependent variables (PLOTY) may be defined on the same graph if each has the same dimensional units. The same parameters are available to the PLOT command as are available to the TABle command. The LABel may be up to 16 alphanumeric characters.

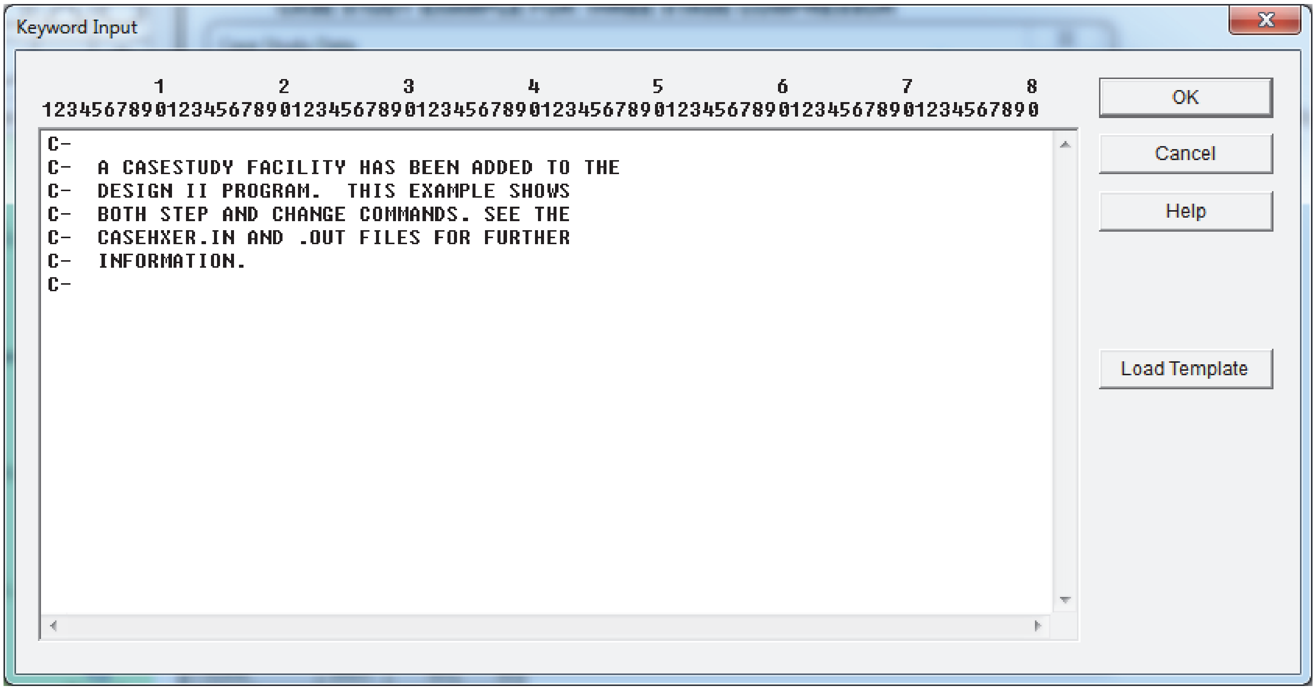


Figure 64: Case Study Data - Keyword Input Dialog (from case1.psd)

3. Open the **Specify** menu and select **Case Study**. The Case Study Data- Keyword Input dialog displays.
4. Enter the desired keyword input, or click the **Load Template** button to load a pre-defined template of keywords. This dialog is similar to the one used for setting Optional General specs: Keyword Input. Refer to the ***Setting Optional General specs: Keyword Input***, ***Creating Keyword Commands***, and ***Editing Keyword Commands*** sections for details.
4. Click the **OK** button when done. The Case Study Data- Keyword Input dialog closes.

Keyword Input

Use **Keyword Input** to enter keyword information not supported by DESIGN II for Windows dialogs; these are commands that appear in the General section of DESIGN II.

DESIGN II for Windows provides dialogs for entering many of the key specifications used in the General section of the DESIGN II input file. However, the program does not currently support all the possible specifications. You can use the Specify-Keyword Input option to enter these commands. Also, a template of unsupported General section keyword commands is provided.

1. Open the **Specify** menu and select **Keyword Input**. The General Data- Keyword Input dialog displays.

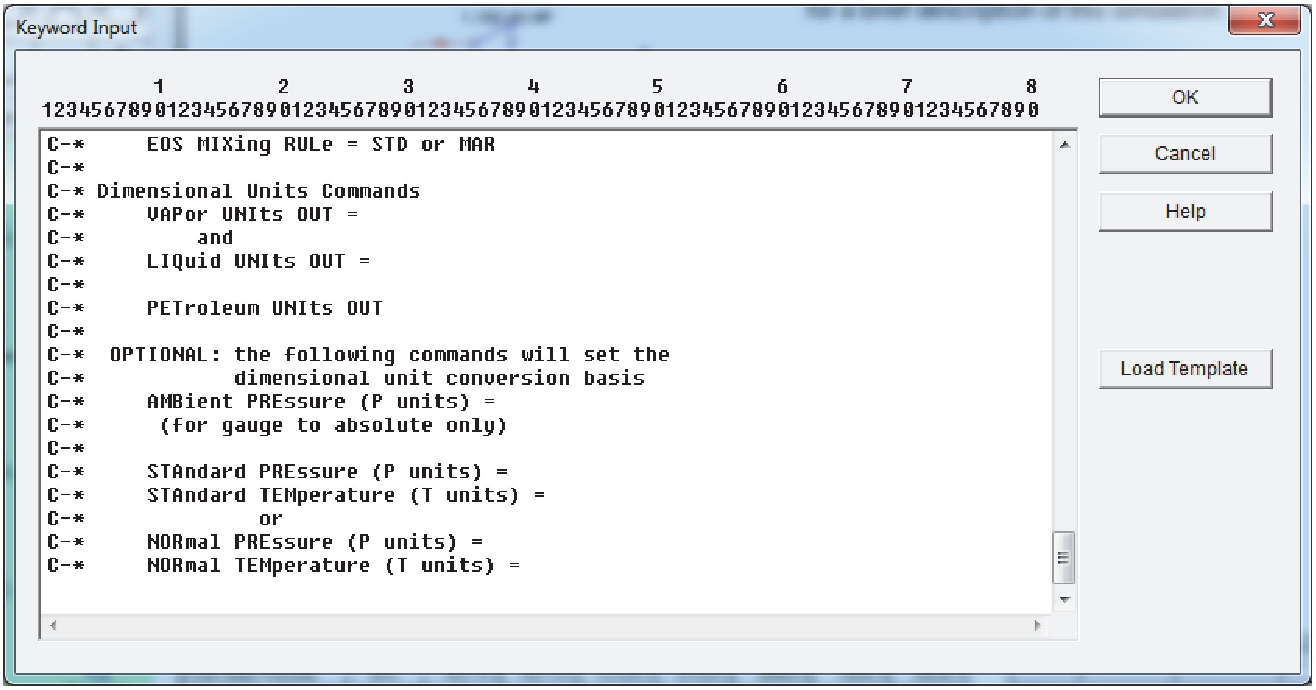


Figure 65: General Data - Keyword Input Dialog (from expander.psd)

2. Type the desired keyword input, or click the **Load Template** button to load a pre-defined template of keywords.

   All lines of the template will begin with the characters 'C-*'. The '*' prevents these lines from appearing in your DESIGN II input. The 'C-' allows you to add comment lines to your DESIGN II input. Equipment modules also have pre-defined keyword templates specific to each module.

   Refer to one of the sections below for details: ***Creating Keyword Commands*** (the procedure to use when you first create keyword commands from a command template) or ***Editing Keyword Commands*** (the procedure to use when you are modifying keyword commands you have already created).

3. Click **OK** when done.

Creating Keyword Commands

When you first use a command-template dialog for an equipment, crude stream, or general flowsheet section, the template will contain all the commands for the particular category not currently supported by DESIGN II for Windows dialogs.

Follow these steps for creating general flowsheet commands for your input:

1. Make sure the General Data- Keyword Input dialog is displayed; then click the **Load Template** button. The template displays.
2. Locate the keyword command you need for your specification.
3. Remove the **'C-*'** using keyboard editing keys (delete, backspace, page up, page down, arrow keys, etc.). You can use the mouse for highlighting text.
4. Edit the command template so it contains your specifications and the correct dimensional units. Be sure to delete the characters specifying the type of dimensional unit (T units, P units, molar Q units/t units, etc.)
5. Remove the **'*'** from the 'C-' for any lines in the template you want to appear in your DESIGN II input file as comments. Lines beginning with a 'C-*' will not appear in your DESIGN II input file.
6. Click the **OK** button when done. The General Data- Keyword Input dialog closes.

Editing Keyword Commands

In the command-template dialog, additional options are provided for editing a previously created set of keyword commands. When you enter the dialog, DESIGN II for Windows will display the text that has been entered previously. If you click on the **Load Templates** button, a dialog with the three additional options described below is displayed. After choosing one of these options, use the procedure described previously for creating keyword commands.

Yes: Use this option if you want to replace the commands displayed with a new copy of the command template.

No: Use this option to append a new command template for this category (equipment, Crude Assay, General data) to the commands displayed. You will most likely need this option if you previously deleted lines from the command template.

Cancel: Use this option to close the dialog, with no action taken.

Using Flowsheet Links

The flowsheet linking technology was used previously for larger flowsheets (generally those with more than fifty unit operations and seventy process streams). This technology is obsolete as of the 9.00 release but maintained for those users' old flowsheets who used this feature.

Flowsheet links in DESIGN II for Windows are implemented at the Windows Graphical User Interface level. The DESIGN II simulator allows up to 9,999 unit operations, 9,999 process streams and 1000 components in any given simulation.

DESIGN II for Windows allows you to set one flowsheet as the primary flowsheet and link one or more dependent flowsheets. There is no practical limit to the number of dependent flowsheets. However, all linked flowsheets must be placed within the same directory.

To create a set of linked flowsheets:

1. Open the flowsheet you want to designate as the primary one.
2. Open the **Specify** menu and select **Flowsheet Links….** The General Data- Flowsheet Links dialog displays.

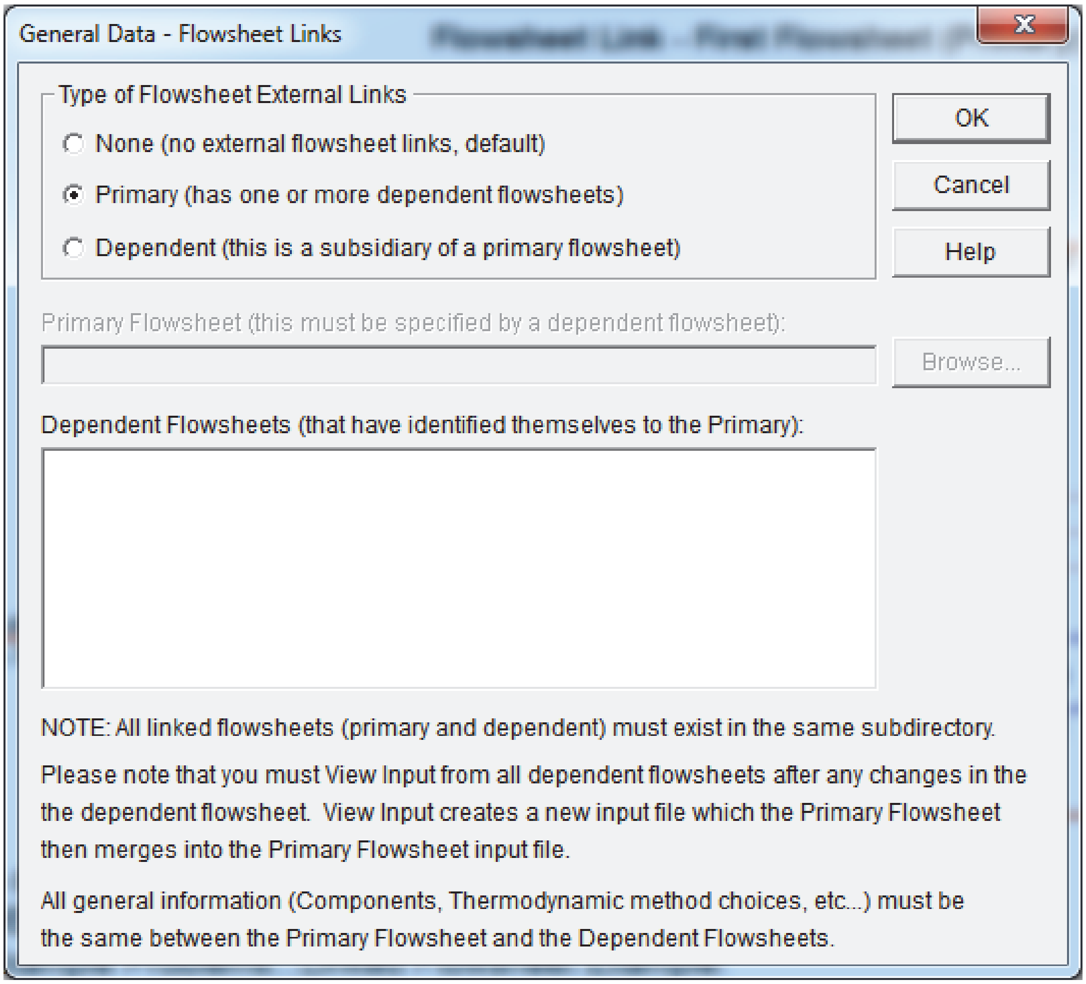


Figure 66: General Data - Flowsheet Links Dialog (from link1.psd)

3. Click on the **Primary** button to set the **Type of Flowsheet External Links**.
4. Click **OK** to confirm your choice. The primary flowsheet is now ready for linking to one or more dependent flowsheets.
5. Next, create or open a flowsheet you want to link to the primary flowsheet (remember, it must be saved in the same directory as the primary flowsheet).
6. Define its Flowsheet Links by opening the **Specify** menu and selecting **Flowsheet Links…** . The General Data- Flowsheet Links dialog displays.
7. Click on the **Dependent** button as the **Type of Flowsheet External Links**.
8. Then, identify the primary flowsheet to which this flowsheet is dependent; click the **Browse...** . button and select the name of the primary flowsheet. You can also type the name of the primary flowsheet in the box labeled **Primary Flowsheet**.
9. Click **OK** to confirm your choice. You have now created a flowsheet linked to the primary flowsheet. Repeat Steps 5-9 to add more dependent flowsheets as necessary.

Specifying Streams for Linked Flowsheets

Streams extending from one flowsheet to the next must be designated as continuation streams. This prevents a validation check that all feed streams on a flowsheet must have temperature, pressure, and flowrate information, since this data will be available during the simulation.

The continuation stream identification is performed using the **Stream** specification dialog. Click on the **check box** at the middle of the Stream dialog to identify a stream as a continuation stream.

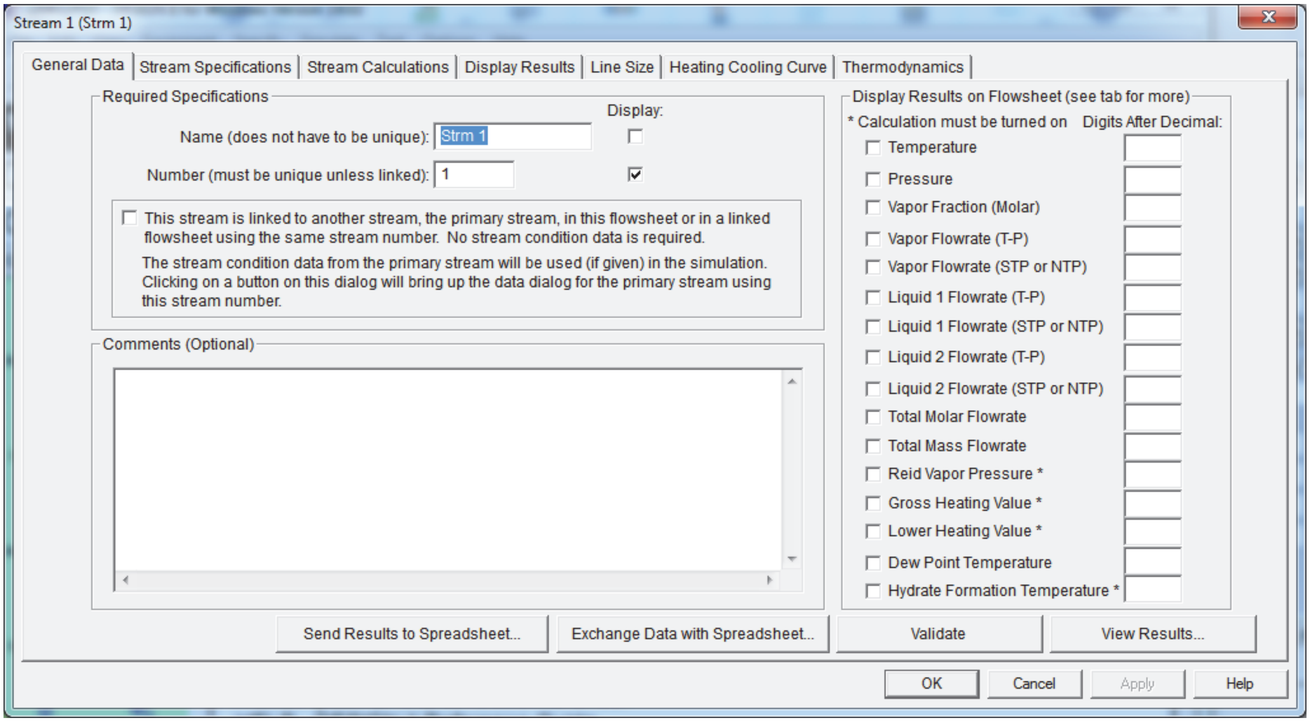


Figure 67: Stream Specification Dialog (from link1.psd)

If you add the equipment and streams to a dependent flowsheet after defining the flowsheet link, DESIGN II for Windows automatically assigns each stream or equipment a unique number in the set of linked flowsheets.

To maintain the continuity of a stream that extends to another flowsheet, you must change the current stream number on the flowsheet where the stream appears as a feed. Simply position the mouse in the box labeled **Number** in the **Stream** dialog box and type in the same number that was used for the product stream from the previous flowsheet.

Simulating Linked Flowsheets

Performing simulations using the linked flowsheets requires two steps:

1. Open each dependent flowsheet and create its input file by opening the **Simulate** menu and selecting **View Input...** . The input file will appear on-screen.
2. Close the input file; then open the next dependent flowsheet, if any, and create its input file.
3. When all the dependent input files have been created, open the primary flowsheet.
   The simulation for the set of linked files is performed from the primary flowsheet. The input files for the dependent flowsheets are merged automatically when you open the **Simulate** menu and select **Execute**.

   NOTE: If your flowsheet includes Optimization or an Inline Fortran Library, this information **must** be included in the primary flowsheet.

   The simulation results can be reviewed from the primary flowsheet or any of the dependent flowsheets using **View Results** for any stream or equipment or by opening the **Simulate** menu and selecting either **View Summary** or **View Results**.

Flowsheet Linking Notes

Unlinking flowsheets

If you want to work with a flowsheet temporarily which has been identified as a dependent flowsheet, click on **Specify** and select the **Flowsheet Links...** option. The General Data - Flowsheet Links dialog appears. Click on the radio button next to the label **None**. The name of the primary flowsheet and any dependent flowsheets will be grayed. Click on **OK** to save your changes.

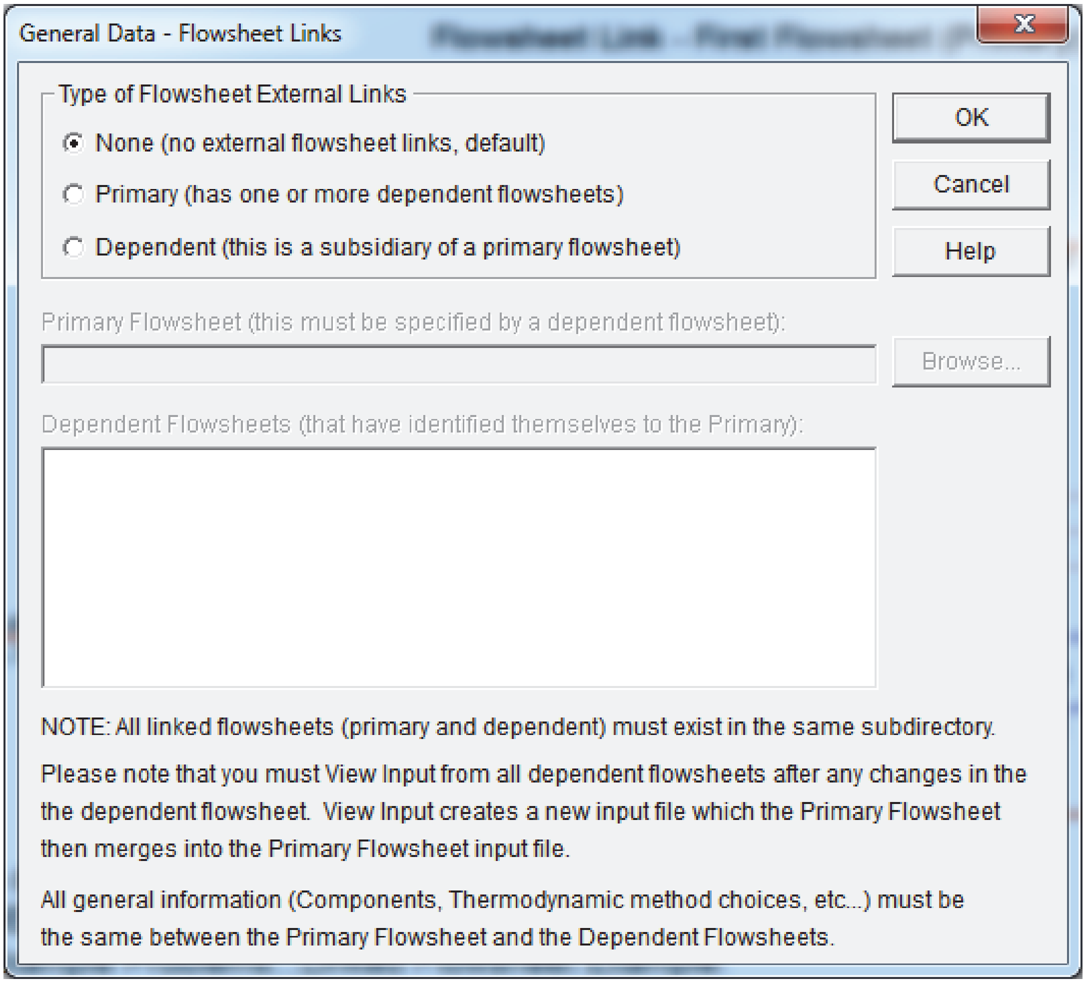


Figure 68: Flowsheet Links (from link1.psd)

The continuation stream designation is removed automatically. You will need to provide temperature, pressure, and flowrate specifications for any feed streams in this flowsheet using the Stream-Basic dialog. See the ***Streams*** chapter for details.

Copying regions

If you copy a region from a linked flowsheet to a new flowsheet using the clipboard, any flowsheet linking is disconnected automatically in the new flowsheet.

Save As

If you use the **Save As...** option from the **File** menu to rename or save a linked flowsheet, the flowsheet links will be disconnected in the new flowsheet. You can relink the newly named flowsheet to an existing linked flowsheet set, but you must ensure all equipment numbers are unique. The linking step will fail if two equipment modules have the same number.

Only those streams that have continued stream checkbox checked are allowed to have duplicate numbers. At the time you identify continuation streams, a message box will pop-up to warn you this stream number has already been used in a flowsheet.

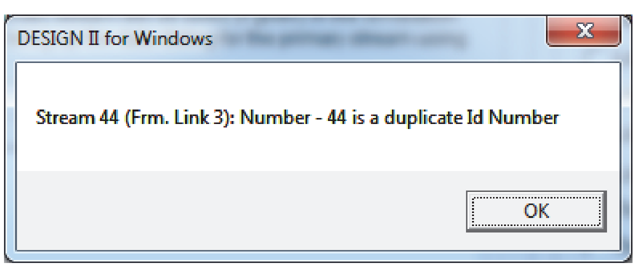


Figure 69: Duplicate Stream Message (from link1.psd)

The name of the flowsheet is provided in the message. Answer **Yes** to use the stream number or **No** to give it a new number.

Chapter 10: Simulation

Viewing Input

You can view the input file for your flowsheet before running a simulation. This optional function works in two ways: it first validates your specifications for the flowsheet; then, if your specifications are accepted, it displays the Input file in its own window on-screen.

To view the input file:

1. Open the **Simulate** menu and select **View Input**. If the file has errors, the DESIGN II for Windows Checklist screen displays. Refer to the section below on correcting errors. If the file is ok, the Input File displays in a new window; this file is named based on your flowsheet drawing, with the extension .IN instead of .PSD. Go to step 2.

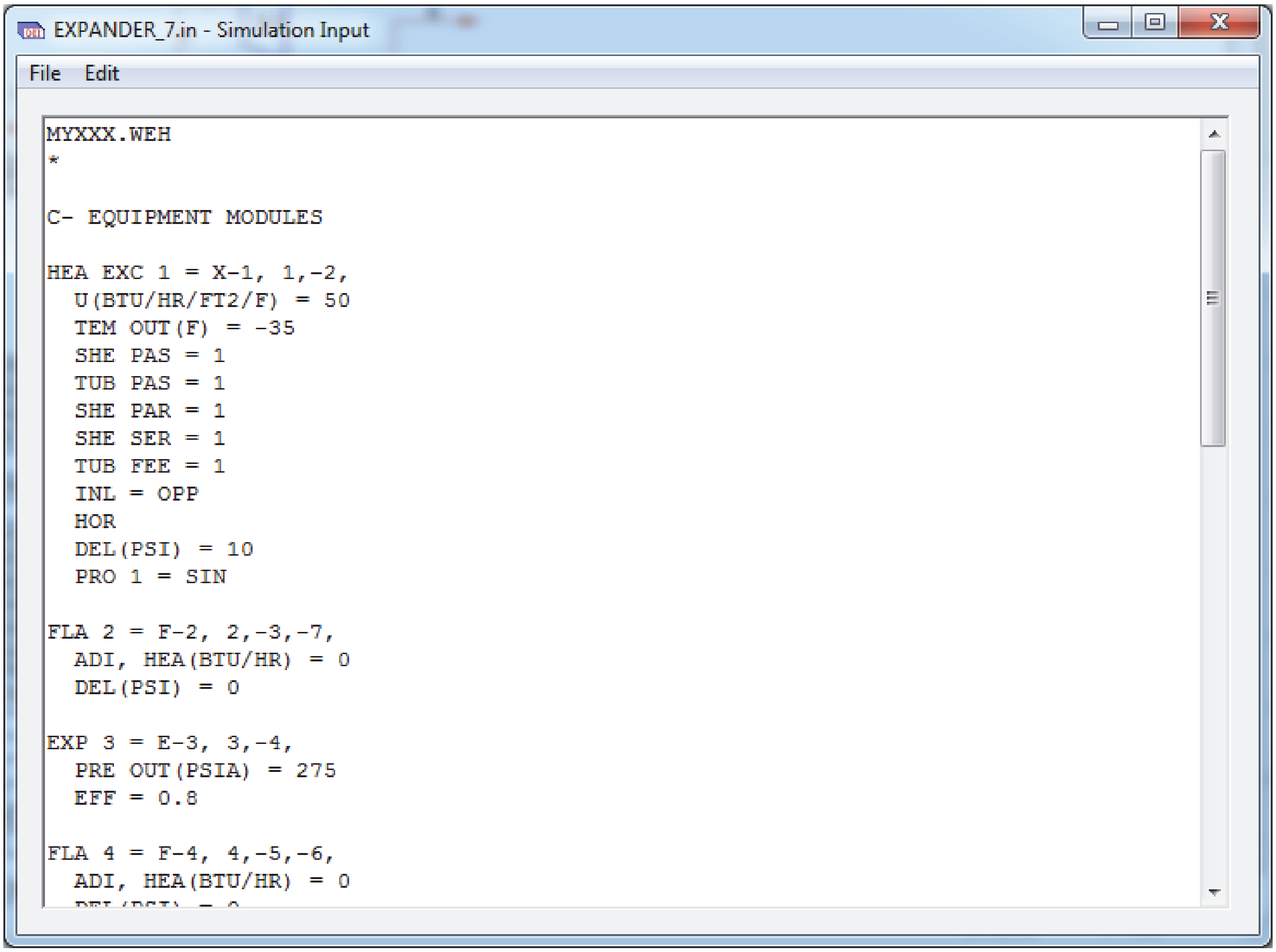


Figure 1: Input File Window (from expander.psd)

2. You can only view the Input file; you cannot make changes to its contents. This restriction is necessary to maintain consistency between your flowsheet specifications and drawing.
   If you are familiar with the DESIGN II keyword input, you might want to review the input and examine the flowsheet specifications you have made. Use the **Scroll bar** or the **Page Down** key to view more material if the input file is longer than one screen.
3. Once you are satisfied with the input file, you can close it and run the simulation.

Correcting Errors

If DESIGN II for Windows detects errors in your specifications, a list of the errors displays.

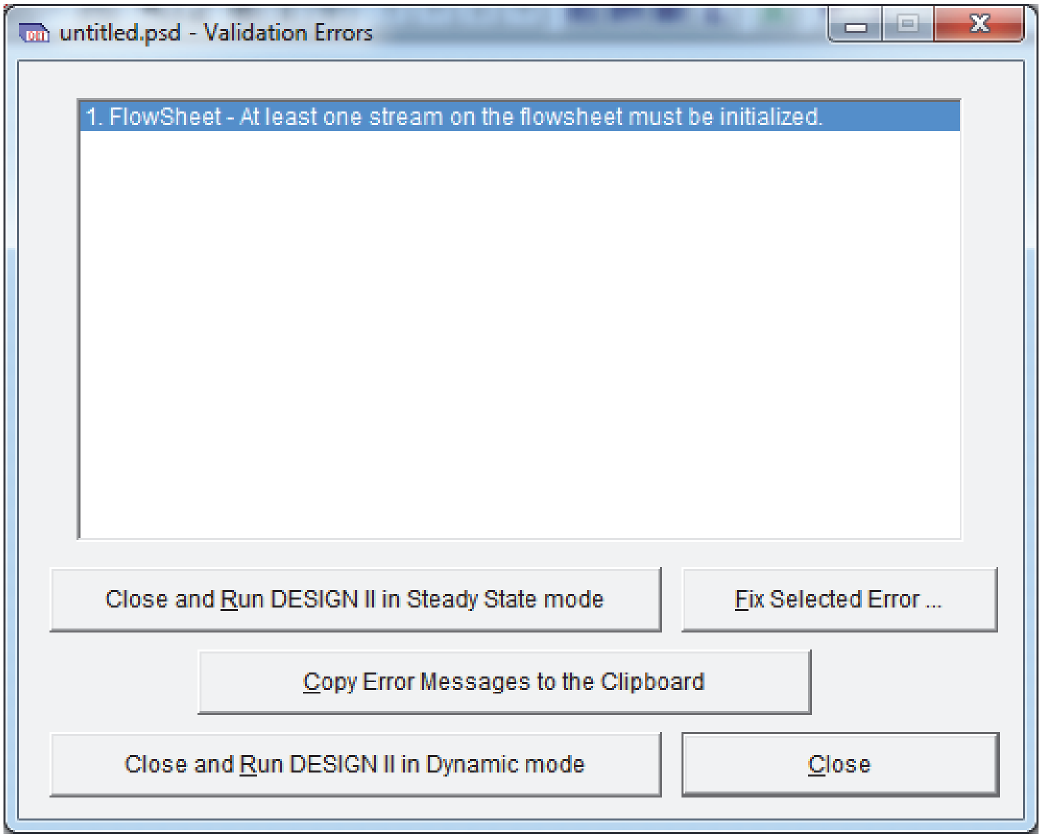


Figure 2: Error Window (from a new flowsheet)

You can correct these errors without going back to the necessary equipment or stream dialogs, by using the following procedure.

1. From the DESIGN II for Windows Checklist screen, select first item in the list of specification problems. Then, click on “Fix Selected Error…”. Use the dialogs that display to correct your specified problems.
2. Close the dialogs.
3. Continue to the next item on the list of specification problems until you have fixed all items.
4. Close the Input File window, ; then select **View Input** from the **Simulate** menu again.
5. If the Input File appears, you have fixed all specification problems; if the DESIGN II for Windows Checklist screen displays again, repeat steps 1-5 again until all problems are corrected.

NOTE: Missing streams and incorrect snap point connections cannot be corrected by double clicking on the error message. You must return to the drawing to make these corrections. When you are finished, click on **Close & Run DESIGN II button** to revalidate your flowsheet and run the simulator.

Checking Input

You can check the input file for your flowsheet before running the simulation. This optional function works in three steps: it first validates your specifications for the flowsheet; then, if your specifications are accepted, it displays the DESIGN II input file on-screen. Finally, Design II for Windows is loaded and your flowsheet is analyzed for the calculation sequence of the equipment; no heat and material balance calculations are performed. The recommended calculation sequence is then reported.

This sequence minimizes the number of recycle streams. Run the Check Input option when evaluating recycle sequences.

To check input: Open the **Simulate** menu and select **Check Input**. Refer to the procedures under ***Viewing Input*** for details on viewing the Input File and correcting errors.

Entering Dynamic Simulation Settings

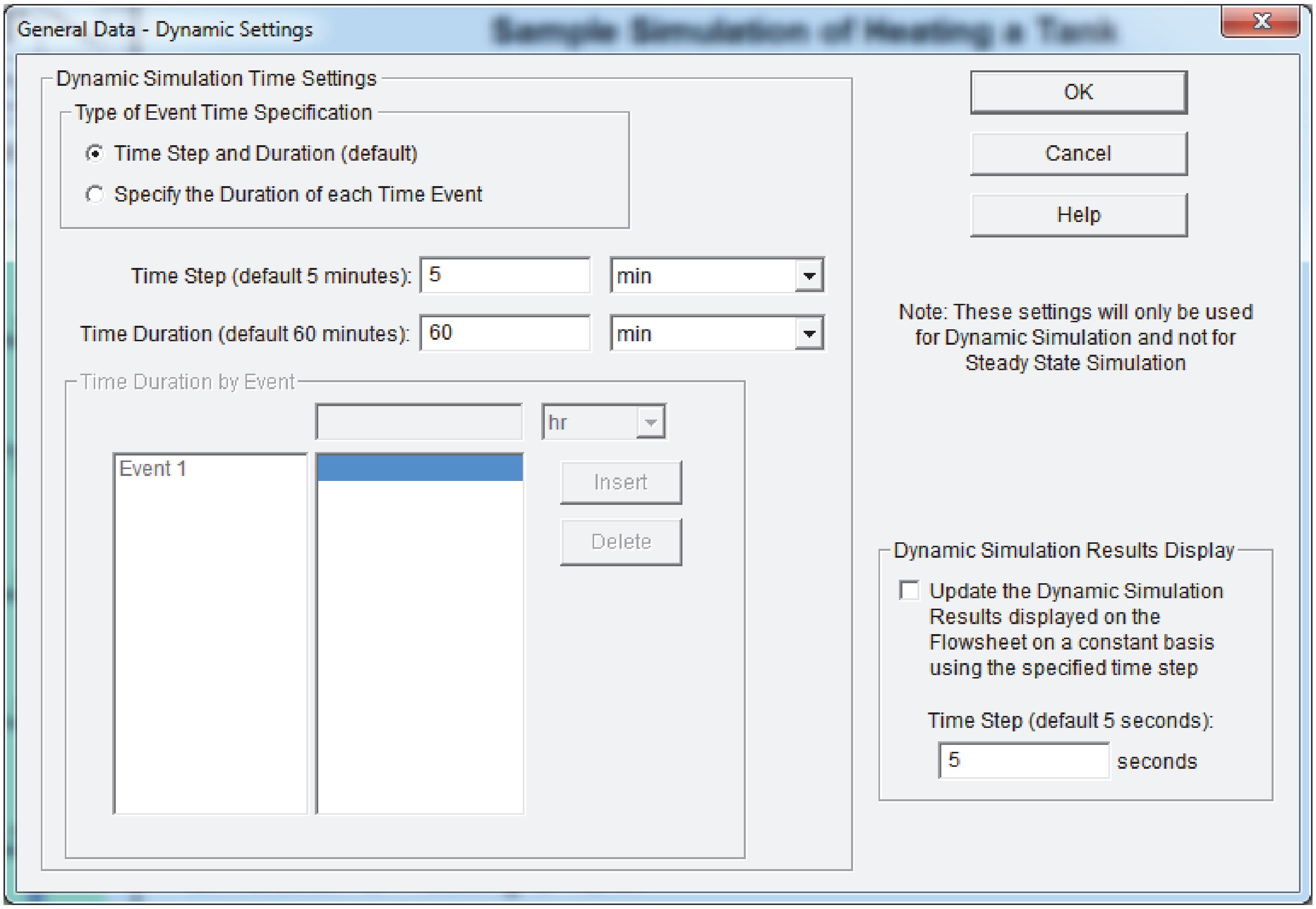


Figure 3: General Data – Dynamic Settings Dialog (from tank.psd)

Open the **Specify** menu and select **Dynamic Settings**. Use these settings when running a dynamic simulation (only Tanks currently support this); these are not used for Steady State simulations. Enter a time step and duration.

Executing

If you have successfully created a flowsheet specification (you verify this by selecting **View DESIGN II Input** under the **Simulate** menu); then you can execute a simulation of your flowsheet using DESIGN II.

After you have finished creating your flowsheet (with all stream and equipment specifications entered), you can run the flowsheet simulation. You can run View Input or Check Input before running the simulation to view/check errors.

NOTE: Save your flowsheet before running a simulation.

You can execute either a dynamic or steady state simulation. Select the dynamic choice when you are ready to run the simulation for your flowsheet, using a dynamic simulation with time duration and interval control; currently, only tank equipment types are supported for dynamic simulations. Time based material and energy reports are included. Open the Specify menu and select Dynamic Settings to enter time duration and interval control settings.

For steady state, select this choice to run a simulation that achieves a steady-state status.

To execute the simulation, select Execute Steady State or Execute Dynamic under the Simulate menu. When DESIGN II starts the simulation, execution messages will appear on the display.

When the simulation is complete, a new dialog will appear on-screen showing the results. Refer to the **Results** chapter for details. You can simulate individual streams or equipment also:

1. To simulate an individual stream using specifications or existing results, make sure the arrow tool is selected from the browser. Click on the stream or equipment to select and highlight it. Then click the right mouse button.

2. From the pop-up menu that appears, choose Simulate this only using Specs or Simulate this only using Results (either stream or equipment). For equipment, you can also choose to run the simulation but use the selected equipment as a stopping point by choosing Simulate (stop after this) from the pop-up menu. The simulation will run and the Simulation Summary dialog appears.

You can simulate individual streams or equipment also:

To simulate an individual stream using specifications or existing results, choose the selection tool (arrow) from the browser. Click on the stream or equipment to select and highlight it. Then click the right mouse button.

From the pop-up menu that appears, choose Simulate this only using Specs or Simulate this only using Results (either stream or equipment). For equipment, you can also choose to run the simulation but use the selected equipment as a stopping point by choosing Simulate (stop after this) from the pop-up menu. The simulation will run and the Simulation Summary dialog appears.

Note: You can save simulation runs under different names by opening the Simulate menu and choosing Save Simulation Results. The General Data Save Current Simulation Results dialog appears. This dialog can also appear automatically before a simulation run.

Using Existing Files

DESIGN II for Windows allows you to use files which you created using a text editor. The Use Existing Files option opens a new window and allows you to load a text input file into a full text editor. You can then make changes, save the file, and run the simulation.

To use existing files:

Open the **Simulate** menu and select **Use Existing Files**. The DESIGN II Input Manager window displays.

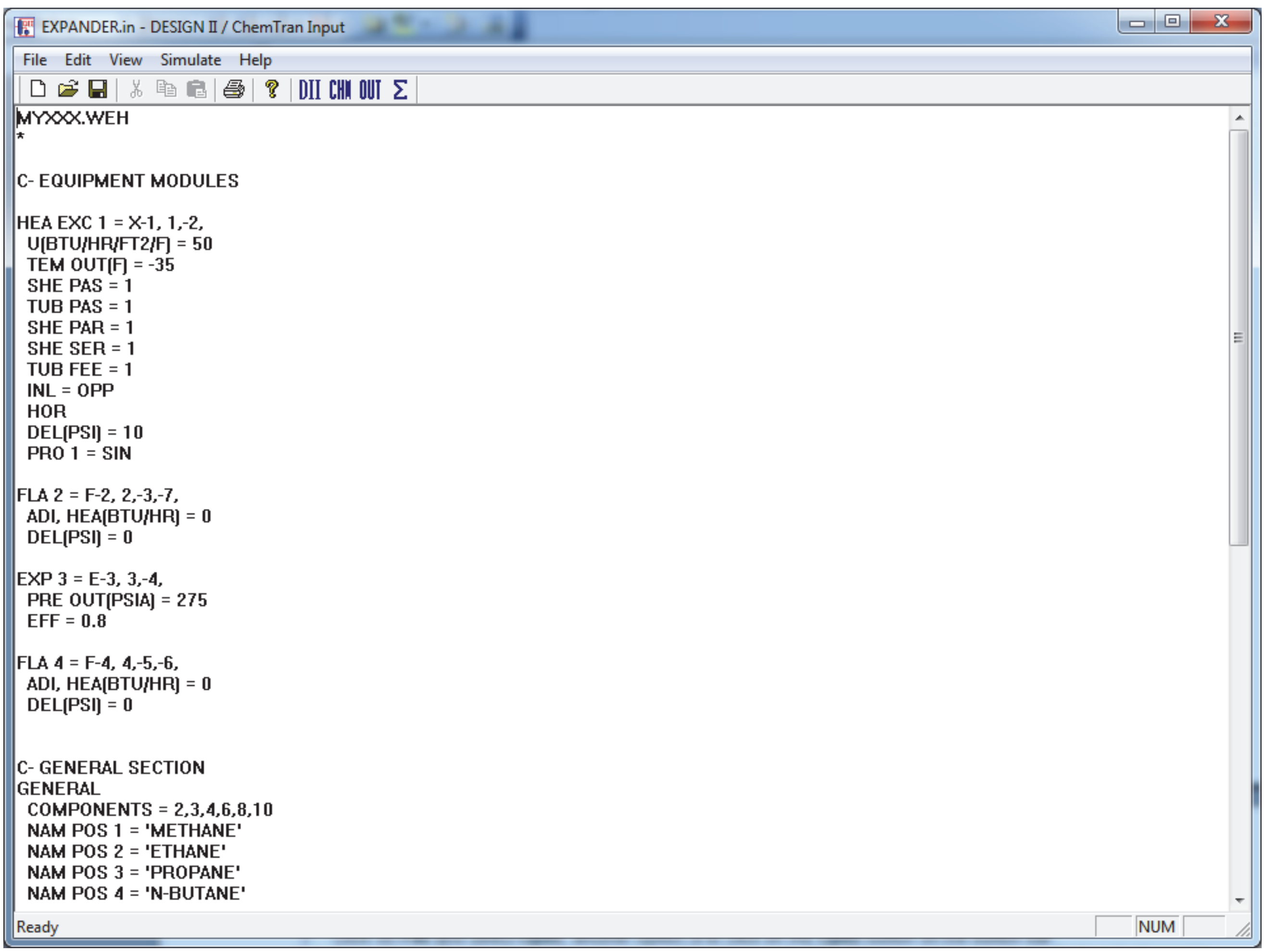


Figure 4: DESIGN II Input Manager Window (using expander.in)

This window provides the following menu options: File, Edit, Search, and Simulate and buttons for: Open, Save, Save As, DESIGN II, ChemTran, and Results.

To use existing files:

1. Click on **File** and select **Open**; another option is to click on the **Open** button on the button bar.
2. A list of input files for the working directory will be displayed. If the file you want to use is not displayed, change to another directory. Select the input file.
3. The input file will be displayed in the edit box. Make any necessary corrections to the file; then click on **File** and select the **Save** option; you may also click on the **Save** button.
4. Click on Simulate and select Execute DESIGN II or click on the DESIGN II button.
5. The Simulation Status window will appear, displaying run-time messages. When the simulation is complete, this window will be dismissed and the Simulation Summary window will appear. See the ***Results*** chapter for details on reviewing the simulation output.
6. Click on **Exit** to dismiss the Simulation Summary window; then click on **File** and select **Exit**.

Exchanging Data with a Spreadsheet

You can transfer data from a spreadsheet to the flowsheet before running a simulation, or transfer the results to a spreadsheet after the simulation is done.Open the **Simulate** menu and select **Exchange Data with Spreadsheet**. The **Exchange Data with Spreadsheet** dialog displays.

You can use this dialog to transfer specification data from an Excel spreadsheet to a flowsheet before running a simulation and to export calculated simulation results from a flowsheet to a spreadsheet. You can control exactly which data items are transferred.

Note: You can transfer data to/from streams, heat exchangers, and air coolers; additional equipment modules will be added in a later version of DESIGN II for Windows.

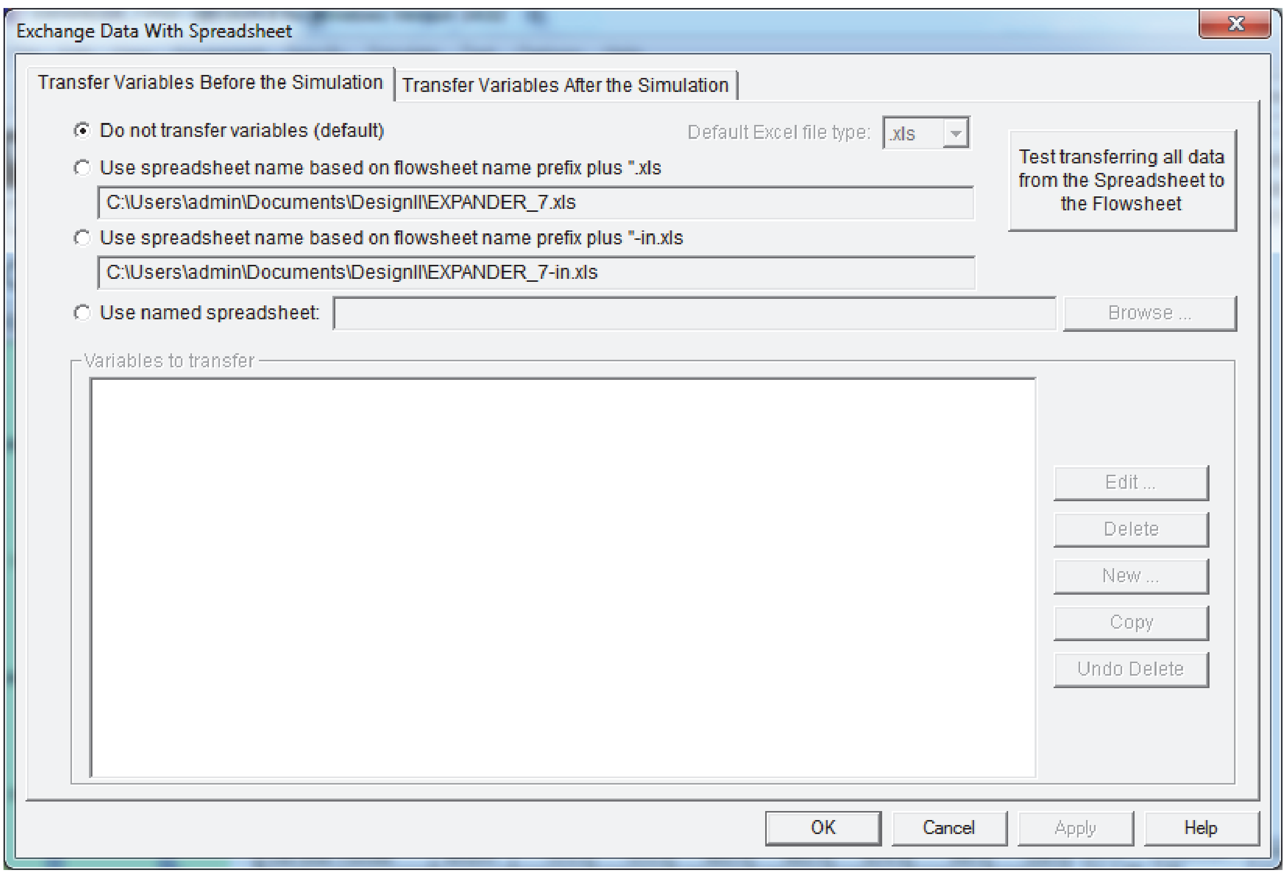


Figure 5: Exchange Data with Spreadsheet dialog

Transfer Variables from Spreadsheet to Flowsheet before the Simulation

Select one of the following choices:

Do not transfer variables (default)

Use spreadsheet name based on flowsheet name prefix plus ".xls": this shows the path where the file must be located, along with the expected filename.

Use spreadsheet name based on flowsheet name prefix plus "-in.xls": this shows the path where the file must be located, along with the expected filename.

Use named spreadsheet: click the Browse button to display a dialog for choosing the drive/directory and file name. When you select the file and click Open on the dialog, the name appears in the Use named spreadsheet field.

Variables to transfer: You can use this option to select specific variables in the spreadsheet to transfer into the flowsheet. Select New to display the **Transfer a Variable from a Spreadsheet** dialog for selecting variables. To edit an existing variable, click on its name in the list and click the Edit button. To remove a variable, click on its name then click Delete. To duplicate a variable so you can go in and make minor edits to create a new variable, click on its name in the list and click Copy.

Transfer Variables from Flowsheet to Spreadsheet after the Simulation

Select one of the following choices:

Do not transfer variables (default)

Use spreadsheet name based on flowsheet name prefix plus ".xls": this shows the path where the file will be located, along with the projected filename.

Use spreadsheet name based on flowsheet name prefix plus "-out.xls": this shows the path where the file will be located, along with the projected filename.

Use named spreadsheet: click the Browse button to display a dialog for choosing the drive/directory and file name. When you select the file and click Open on the dialog, the name appears in the Use named spreadsheet field.

Variables to transfer: You can use this option to select specific variables in the flowsheet to transfer into the spreadsheet. Select New to display the **Transfer a Variable from a Flowsheet dialog** for selecting variables.

Test transferring all data from the Spreadsheet to the Flowsheet/Test transferring all data from the Flowsheet to the Spreadsheet

Once you've set up a variable transfer (either from the spreadsheet or from the flowsheet), click the appropriate button to test the data transfer before you actually perform the final data transfer.

Transfer a Variable from a Spreadsheet to the Flowsheet Before Simulation

You can use this dialog to select specific variables to transfer from an Excel spreadsheet to the flowsheet before running a simulation and to export specific calculated simulation results from a flowsheet to a spreadsheet.

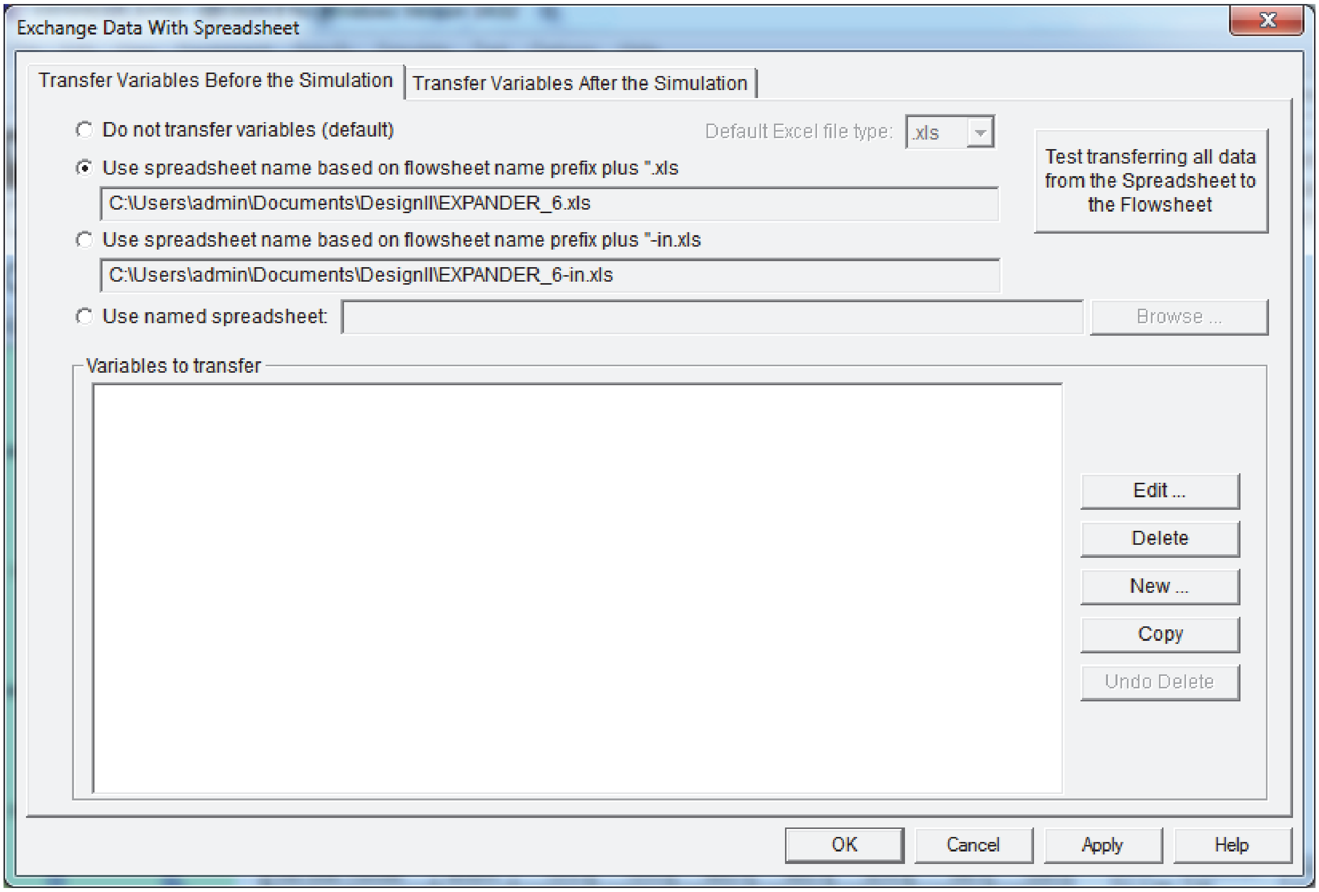


Figure 6: Transfer a Variable from a Spreadsheet to the Flowsheet before the Simulation dialog

Transfer a Variable from Spreadsheet to Flowsheet before the Simulation

Select one of the following choices:

Do not transfer variables

Use spreadsheet name based on main transfer variables dialog (default) this shows the path where the file must be located, along with the expected filename.

Use named spreadsheet: click the Browse button to display a dialog for choosing the drive/directory and file name. When you select the file and click Open on the dialog, the name appears in the Use named spreadsheet field.

Transfer Variable from this location in the Spreadsheet

Column (Excel 95 to 2003: A to IV; Excel 2007: A to XFD): based on the version of Excel you are using, enter the appropriate column letter that contains the variable.

Row (Excel 95 to 2003: 1 to 65,536; Excel 2007: 1 to 1,048,576): enter the appropriate row number that contains the variable.

Sheet name: Enter the name of the worksheet that contains the variable.

Get the Row number, Column number, and Sheet name from the currently selected cell in this Spreadsheet: if you have the desired spreadsheet file open in Excel, click on the variable cell; then click this button to automatically populate the Column, Row, and Sheet name fields on this dialog.

Transfer Variable to this location in the Flowsheet

Select Stream or Equipment.

Number:Name: Select the desired object (either stream name or equipment name) from the list. For equipment, you can only select a heat exchanger or air cooler.

Specification: After selecting the desired object, open the list and select the specification to which you want to transfer the variable from Excel.

Note: Vectored specifications will be read vertically from the same column in the spreadsheet, starting with the cell address (column, row, sheet) that you entered above.

Test Transferring the Variable from Spreadsheet to Flowsheet

Once you've set up a variable transfer, click the appropriate button to test the variable data transfer before you actually perform the final transfer.

View Properties for this Stream or Equipment

Once you've selected the desired object (stream or equipment), click this button to open the corresponding properties dialog.

Transfer a Variable from a Flowsheet to the Spreadsheet after Simulation
You can use this dialog to select specific variables to transfer from the flowsheet after running a simulation to a spreadsheet.

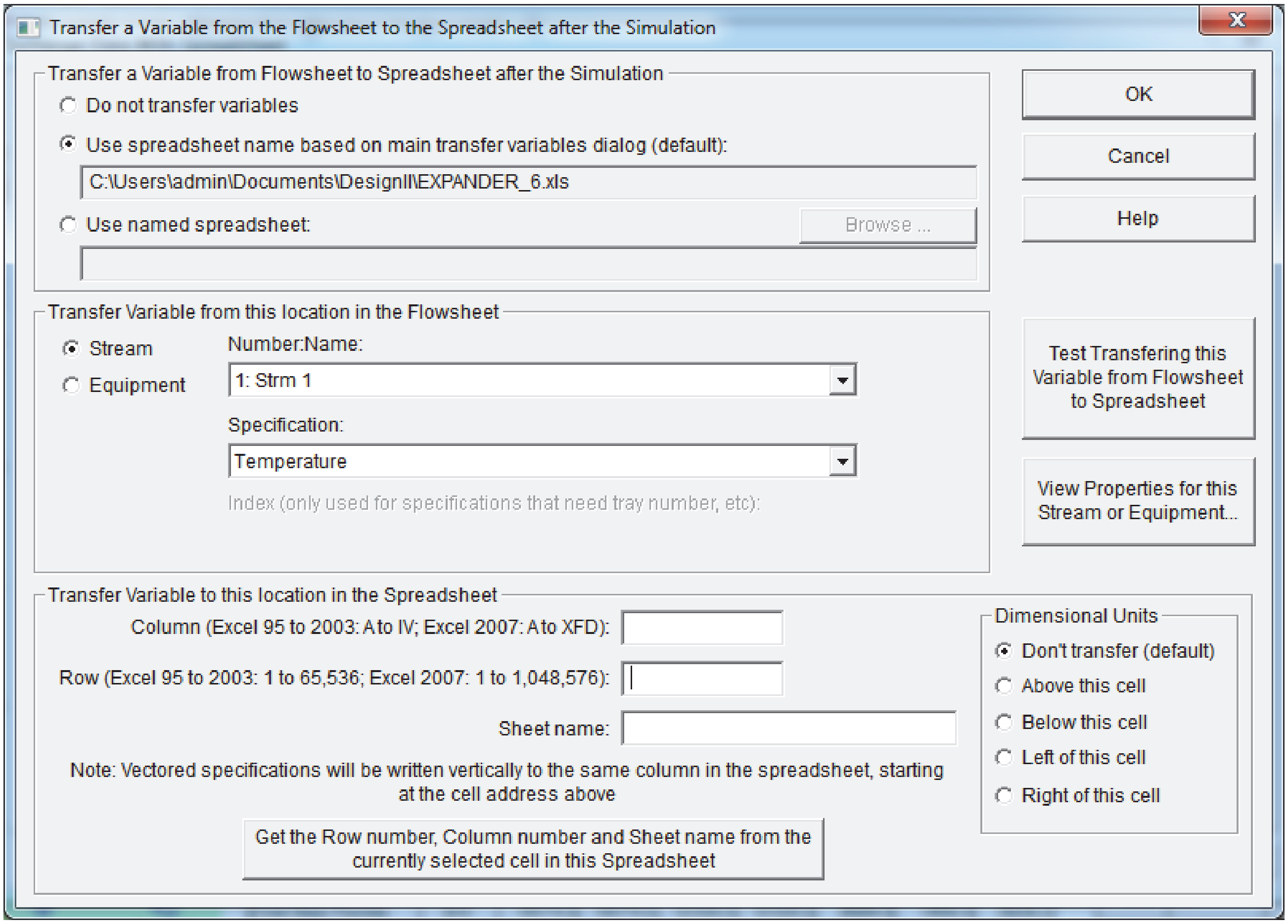


Figure 7: Transfer a Variable from the Flowsheet to the Spreadsheet after the Simulation dialog

Transfer a Variable from Flowsheet to Spreadsheet after the Simulation

Select one of the following choices:

Do not transfer variables

Use spreadsheet name based on main transfer variables dialog (default) this shows the path where the file will be located, along with the projected filename.

Use named spreadsheet: click the Browse button to display a dialog for choosing the drive/directory and file name. When you select the file and click Open on the dialog, the name appears in the Use named spreadsheet field.

Transfer Variable from this location in the Flowsheet

Select Stream or Equipment.

Number:Name: Select the desired object (either stream name or equipment name) from the list. For equipment, you can only select a heat exchanger or air cooler.

Specification: After selecting the desired object, open the list and select the specification you want to transfer to Excel.

Note: Vectored specifications will be written vertically to the same column in the spreadsheet, starting with the cell address (column, row, sheet) that you will enter below.

Transfer Variable to this location in the Spreadsheet	Column (Excel 95 to 2003: A to IV; Excel 2007: A to XFD): based on the version of Excel you are using, enter the appropriate column letter to which you want to write the variable. Row (Excel 95 to 2003: 1 to 65,536; Excel 2007: 1 to 1,048,576): enter the appropriate row number to which you want to write the variable. Sheet name: Enter the name of the worksheet to which you want to write the variable. Get the Row number, Column number, and Sheet name from the currently selected cell in this Spreadsheet: if you have the desired spreadsheet file open in Excel, click on the variable cell; then click this button to automatically populate the Column, Row, and Sheet name fields on this dialog. Dimensional Units: select how to handle the transfer of dimensional units for the variable data to the spreadsheet based on the column/row/sheet name you entered: Don't transfer (default) Above this cell Below this cell Left of this cell Right of this cell
Test Transferring this Variable from Flowsheet to Spreadsheet	Once you've set up a variable transfer, click this button to test the variable data transfer before you actually perform the final transfer.
View Properties for this Stream or Equipment	Once you've selected the desired object (stream or equipment), click this button to open the corresponding properties dialog.

Chapter 11: Results

Viewing a Summary

After running your flowsheet simulation, a summary of the output file appears on-screen automatically. The summary page provides a message indicating if a solution has been reached. You can also view specific pages of the output file, or export results to a Microsoft Excel spreadsheet (you must purchase Microsoft Excel separately) from this window.

1. Open the **Simulate** menu and select **Check Input**. The simulation runs and then the Simulation Summary dialog displays.

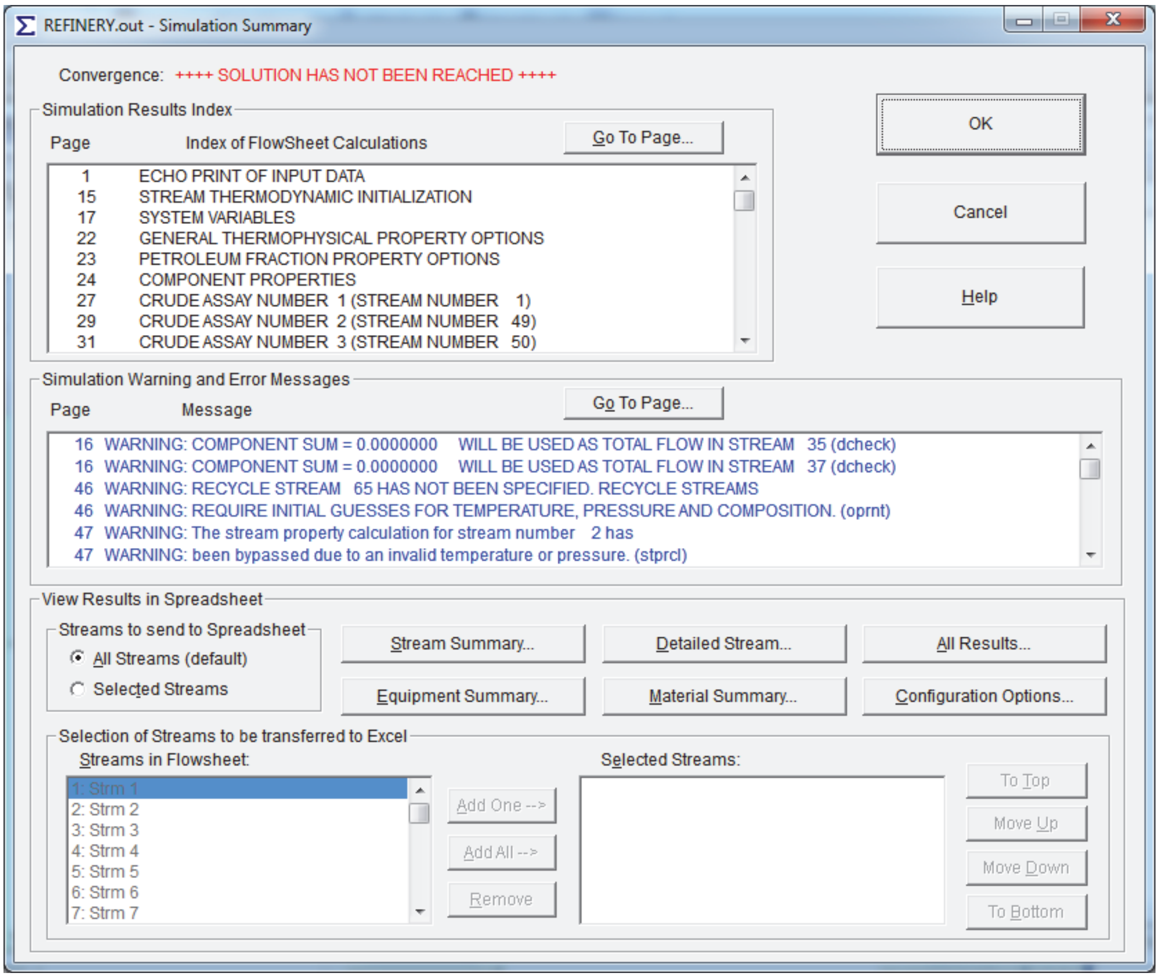


Figure 1: Summary Dialog (from refinery.psd)

2. You can use the following Simulation Summary options:

   When DESIGN II has finished executing, a new window is displayed showing optional ways of reviewing your output. The output will be placed in a file with the extension .OUT and the same name as your flowsheet file (.PSD extension) and your input file (.IN extension). This summary provides you several options for handling the output file.

   Simulation Results Index: The upper group box in the View Summary page is titled Simulation Results Index. The box lists the various section of your output file with the appropriate page number. By double-clicking on one of the entries (or clicking on the Go To Page… button), such as STREAM SUMMARY or EQUIPMENT SUMMARY, the text output file will be opened to hat page for your review. Once the text output file is open, you have all of the powerful edit techniques which you receive when choosing the View Results option.

   Simulation Warning and Error Messages: The middle group box in the View Summary page is titled Simulation Warning and Error Messages. The box lists the WARNING, ERROR and NOTE: messages in your output file with the appropriate page number. By double-clicking on one of the entries (or clicking on the Go To Page… button), the text output file will be opened to that page for your review. Once the text output file is open, you have all of the powerful edit techniques which you receive when choosing the View Results option.

View Results in Spreadsheet: Design II for Windows provides automatic links to the Excel spreadsheet. The lower group box on the View Summary Page is titled "View Results in Spreadsheet" and contains five buttons which load output data into Microsoft Excel. By selecting one of these buttons, Design II for Windows automatically opens Excel and loads the data into individual pages in one book. The book is formatted so your information is neatly displayed with appropriate column width, spacing, and text size. The titles for each page are large and bright. The tabs for each page are shown at the bottom of the screen and are labeled according to the information on the page.

The file loaded into Excel will have the same name as your PSD file but will have the extension XLS. You have the full capabilities of Excel available including graphing functions. You must use the File-Save command if you wish to save the file to your disk.

The five buttons shown on the View Summary Page are labeled: Stream Summary, Equipment Summary, Material Summary, All Results, and Detailed Stream. By choosing one of the buttons, Design II for Windows will load the appropriate information into the spreadsheet. All Results loads the Stream Summary, Equipment Summary and Material Summary into the spreadsheet.

3. Click **Exit** when done with the Simulation Summary dialog.

Viewing Results

After running the flowsheet simulation, you can review your DESIGN II output file.

1. Open the **Simulate** menu and select **View Results**. The Results window displays with your output file, named using your flowsheet name with the extension ***.OUT*** added.

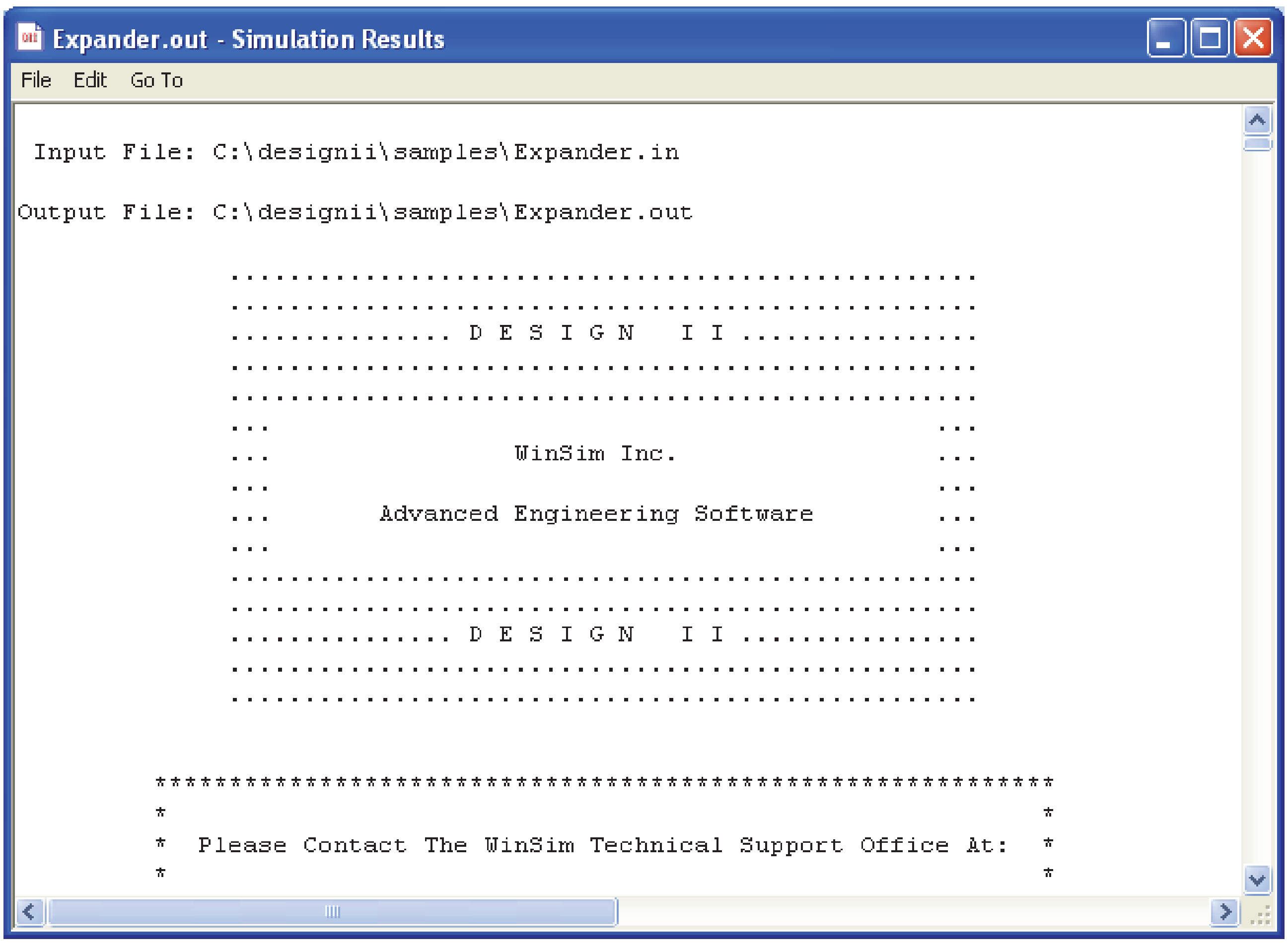


Figure 2: Results Window (from expander.psd)

2. Review the file. You have the following options available:

File: Open another output file, print the current file, or exit from the Results window.
Edit: Copy contents of the output file, which you have highlighted, to the clipboard (for pasting into another application); use **Select All** to highlight the entire contents of the output file.
Search: Find and display a string of text, which you type into a search dialog, in the output file.
Page Up: Click this button to move to the preceding page in your output file.
Page Dn: Click this button to move to the next page in your output file.

Go To: Click this button to go directly to the page number of the output you want to display. After you click this button, a dialog appears which you use to enter the number of the page you want to display. You can use this button in conjunction with the Index button to quickly locate sections of the output file.
Index: Click this button to display the index for the output file. This index lists page numbers for significant sections of the output file. You can use this button in conjunction with the Go To button to locate sections of the output file, such as the Stream Summary.
Scroll Bar: Drag the scroll box up or down on the scrollbar to view more of the output file.

3. When done viewing the file, open the **File** menu and select **Exit**. The Results window closes.

Transferring Data

You can transfer results data to other software programs, such as Microsoft Word (you must purchase this separately).

To transfer data to a Microsoft Excel spreadsheet, use the procedure outlined under the ***View Summary*** section. Once the data is loaded into the spreadsheet, you have the full capabilities of Excel available including graphing and /charting functions.

If you select Configuration Options from the Simulation Summary screen, you can select where to place the data transfer results from an output file, which streams to transfer, and other options.

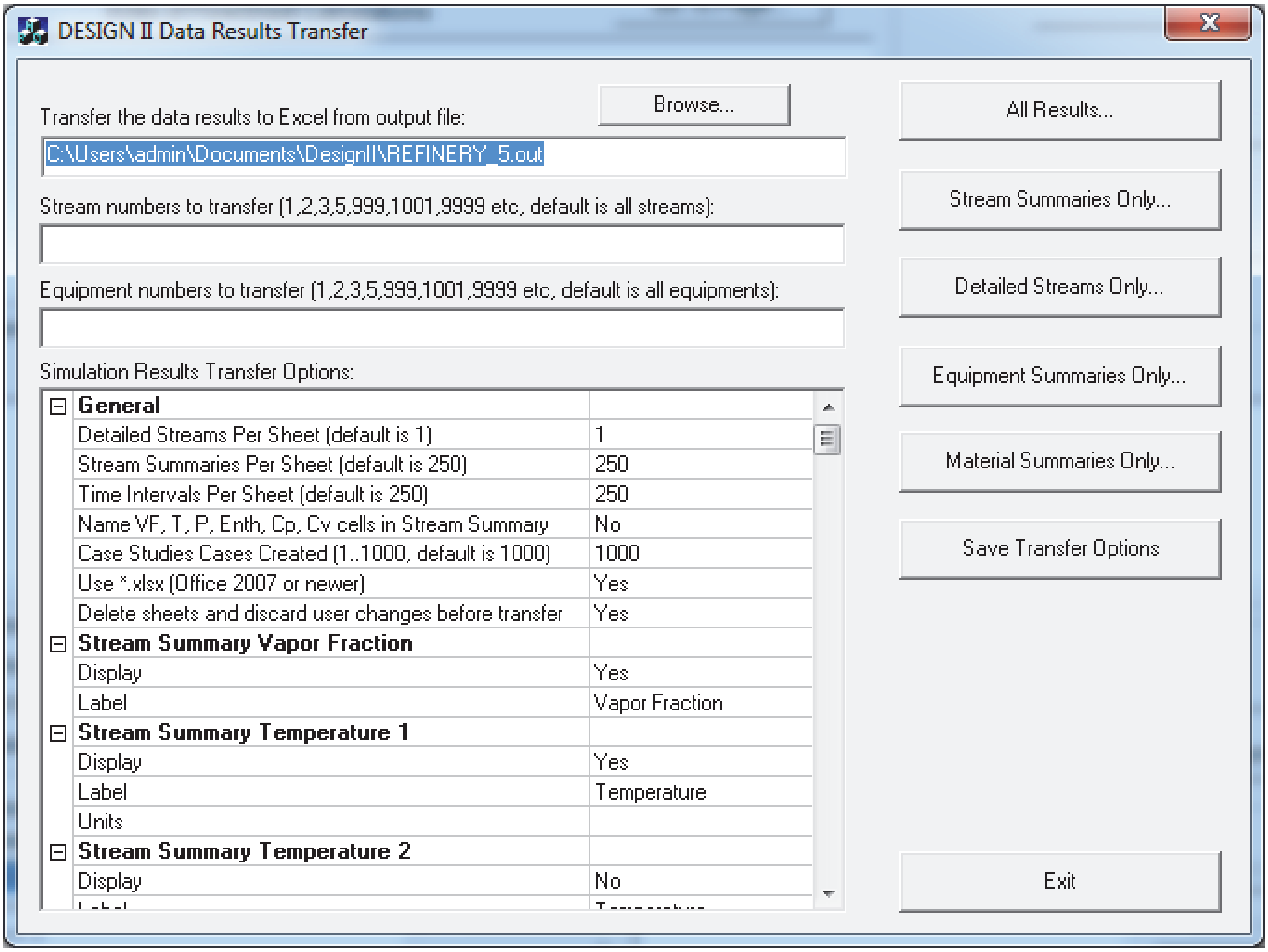


Figure 3: Data Results Transfer Window (from refinery.psd)

To transfer data to word processing programs for generating reports, follow these steps:

1. Make sure the word processing program to which you want to transfer data is running.
2. Open the **Simulate** menu and select **View Results**. The Results window displays your output file.
3. Locate the desired information you want to transfer. Use **Search, Go To, Page Up/Down**, etc., to locate this information.
4. With the desired information displayed in the Results window, place the cursor at the start of the text you want to transfer. Hold down the mouse button and drag the highlight down the page until you reach the last of the information you want to transfer; then, release the mouse button. The text is highlighted.
   To select all information from the current page of the output file for transferring, open the **Edit** menu and choose **Select All**. All text on the page is highlighted.

5. Open the **Edit** menu and select **Copy.** The highlighted text is copied into the Windows clipboard. You can now switch to your word processing program and paste the text using the standard **Edit/Paste** function.

Stream Data Box

You can display all or some of the stream results on the drawing in a stream data box if you desire. Pull down the Simulate menu and select Stream Data Box… The Import Stream Data Dialog will popup and allow you to select which streams you want displayed on the drawing in the box.

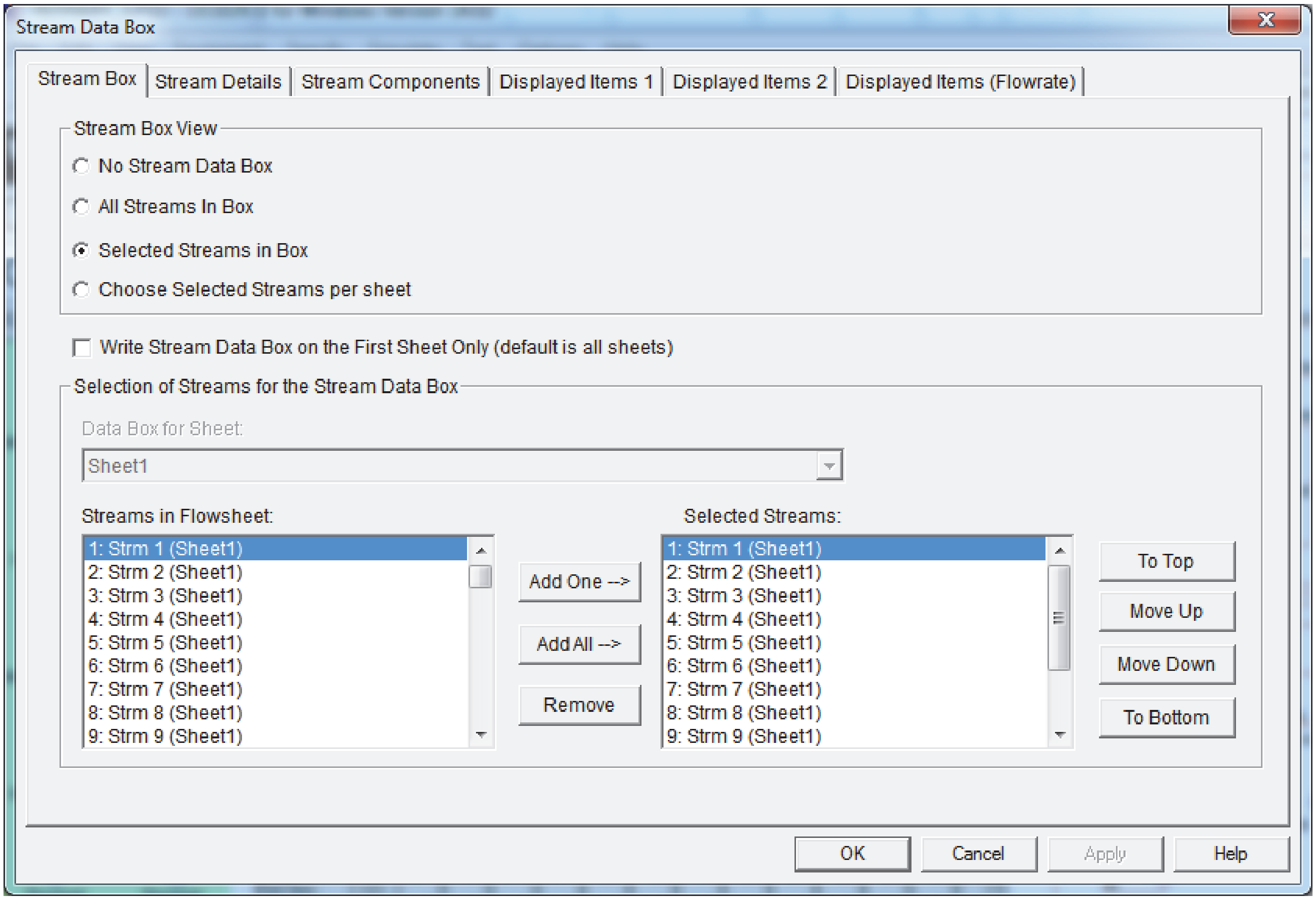


Figure 4: Stream Data Box Dialog (from refinery.psd)

Please note that the information in the stream data is read from the simulator output file. If the simulator output file does not exist the stream data box will be empty.

You can change the formatting and placement of the Stream Data Box by clicking on the Stream Details tab. You can change the Stream Data Box placement, text height, row height and number of rows of streams in the stream data box.

You can change the set of items in the Stream Data Box by clicking on the Display Items tab or Display Items (Continued) tab. You can change the display of the items, the displayed row label and number of digits of precision Stream Data Box placement. On some of the items, you can also change the displayed dimensional units.

By default, all sheets will have stream boxes on them (with the streams on each sheet in each box). However, you can change this so that only the first sheet has the stream box on it by clicking the check box.

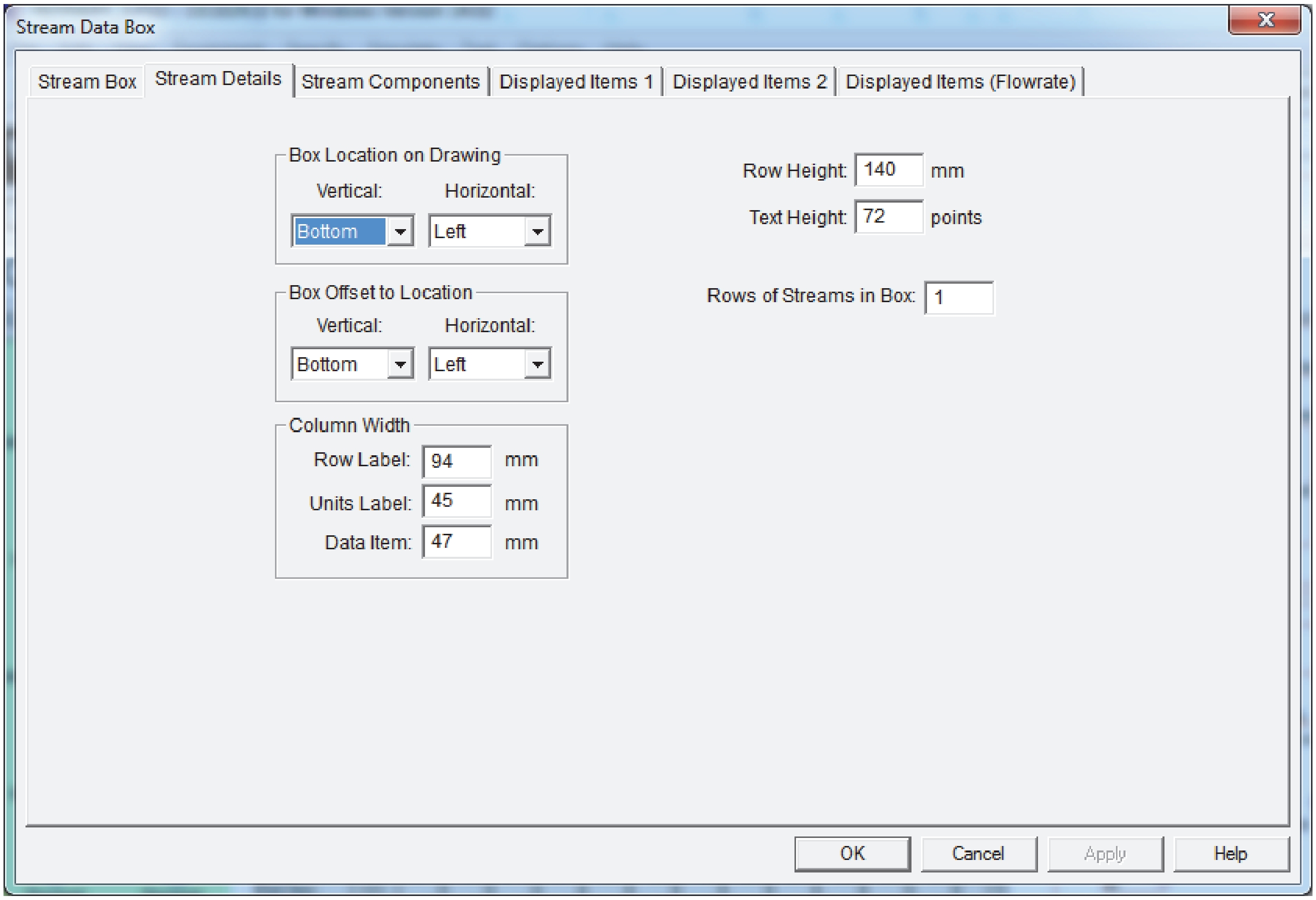


Figure 5: Stream Details Tab (from refinery.psd)

This dialog is to control the view of stream results data on the flowsheet drawing.

Data Item	Description
Box Location on Drawing	Controls location of the stream box on the drawing, Horizontal - Left, Center, Right, Vertical - Bottom, Center, Top
Box Offset to Location	Controls the offset of drawing the stream box to the location Horizontal - Left, Center, Right, Vertical - Bottom, Center, Top
Row Height	The height of each row (in mm) of the stream data box
Text Height	The height of all text (in points) in the stream data box. All text is drawn using the Arial font.
Rows of Streams in Box	Enter the number of streams to display in the Stream Data Box.
Column Width	Controls the width of the first column (row label), the dimensional unit column and the subsequent columns (the selected streams)

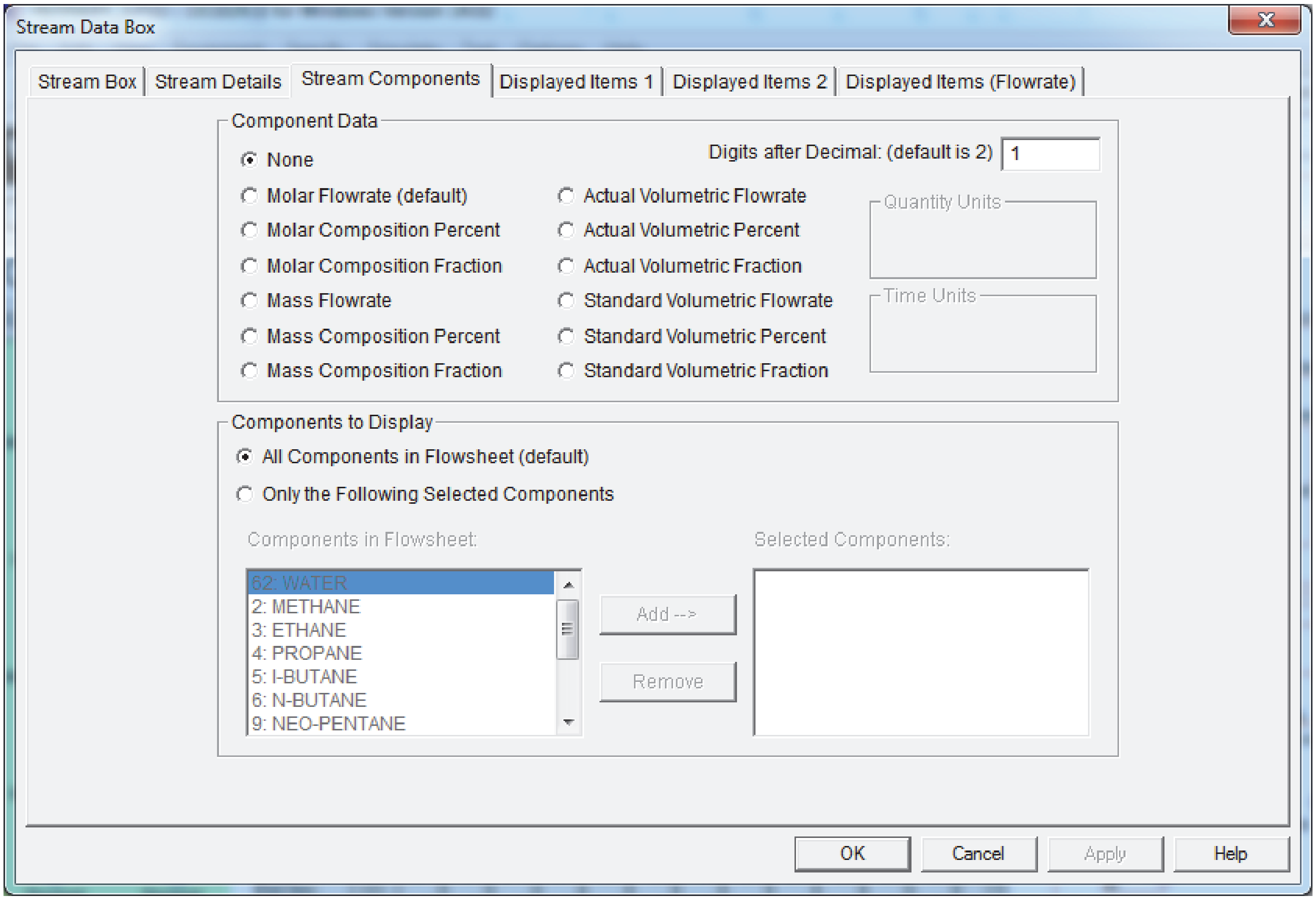


Figure 6: Stream Components Tab (from refinery.psd)

This tab dialog allows you to set display options for the **Stream Data Box**.

Data Item	Description
Component Data:	Set the type of the component data to display, along with the number of digits to use: **None** **Molar Flowrate** **Molar Composition Percent** **Molar Composition Fraction** **Mass Flowrate** **Mass Composition Percent** **Mass Composition Fraction** **Actual Volumetric Flowrate** **Actual Volumetric Percent** **Actual Volumetric Fraction** **Standard Volumetric Flowrate** **Standard Volumetric Percent** **Standard Volumetric Fraction**
Digits after Decimal (default is 2)	Enter the number of digits you want to display after the decimal for the component data.
Components to Display:	Choose to display all components or only certain selected components in which to add the chosen molar/mass data to the bottom of the stream data box. If you select Only the Following Selected Components, you can choose flowsheet components from the list and click Add to move them to the Selected Components list. To remove a component, click on it in the Selected Components list and click the Remove button.

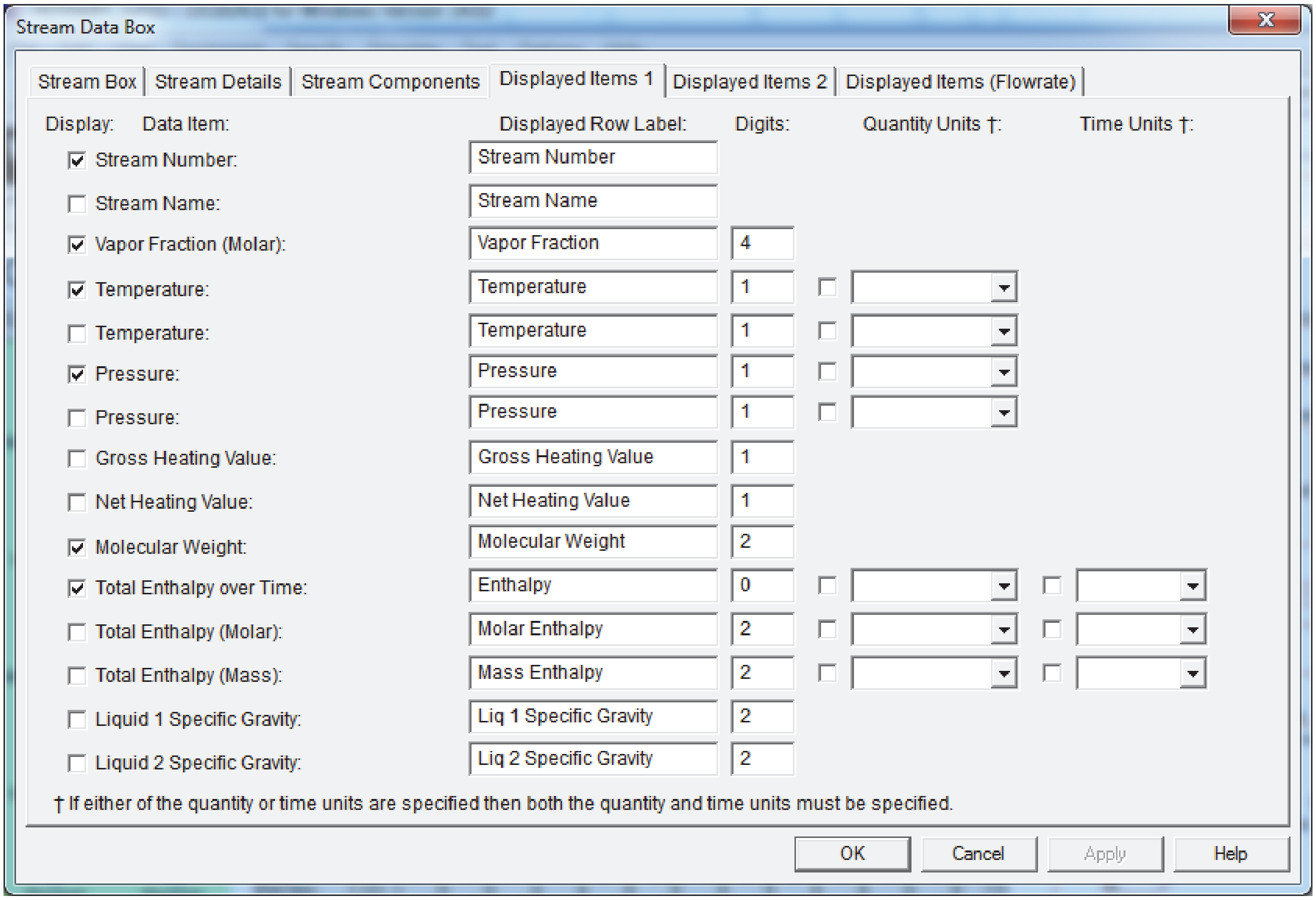


Figure 7: Displayed Items 1 Tab

You can display the

- Stream Number
- Stream Name
- Vapor Fraction
- Temperature (2)
- Pressure (2)
- Gross/Net Heating Values
- Molecular Weight
- Flowrate Enthalpy
- Total Molar Enthalpy
- Total Mass Enthalpy
- Liquid 1 Specific Gravity/Liquid 2 Specific Gravity

Data Item	Description
Display	Checkbox that controls whether or not a row is displayed in the stream box
Data Item	Data item that will be displayed in a row
Displayed Row Label	The text of the data item that will be displayed in the first column of a row
Digits	Controls the number of decimals displayed after the decimal point for numeric values
Units	Controls the displayed quantity and/or time units

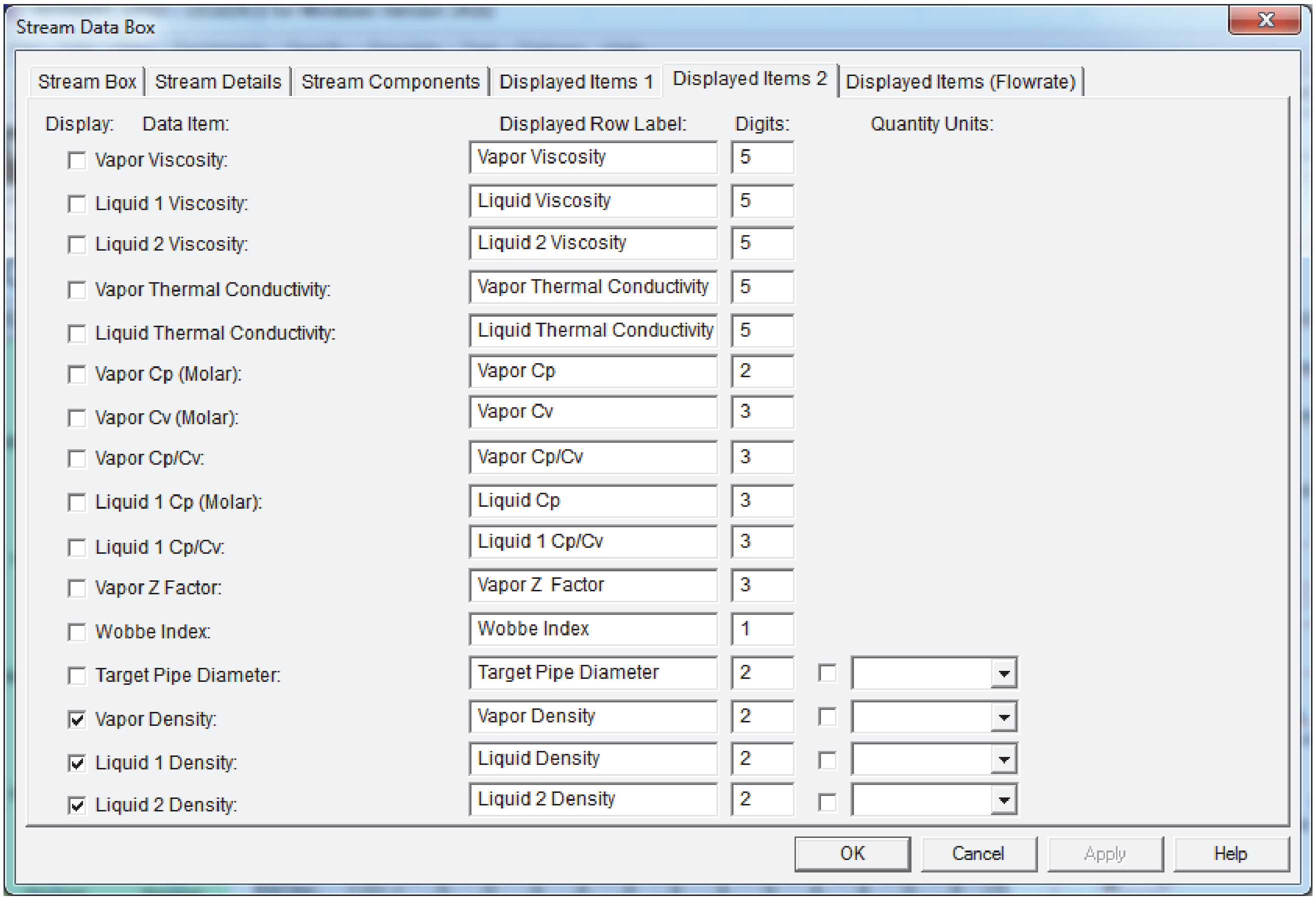


Figure 8: Displayed Items 2 Tab

You can display the

- Vapor Viscosity
- Liquid 1 Viscosity
- Liquid 2 Viscosity
- Vapor Thermal Conductivity
- Liquid Thermal Conductivity
- Vapor Cp
- Vapor Cv
- Vapor Cp/Cv
- Liquid 1 Cp
- Liquid 1 Cp/Cv
- Vapor Z Factor
- Wobbe Index
- Target Pipe Diameter (select a Quantity unit)
- Vapor Density
- Liquid 1/2 Density

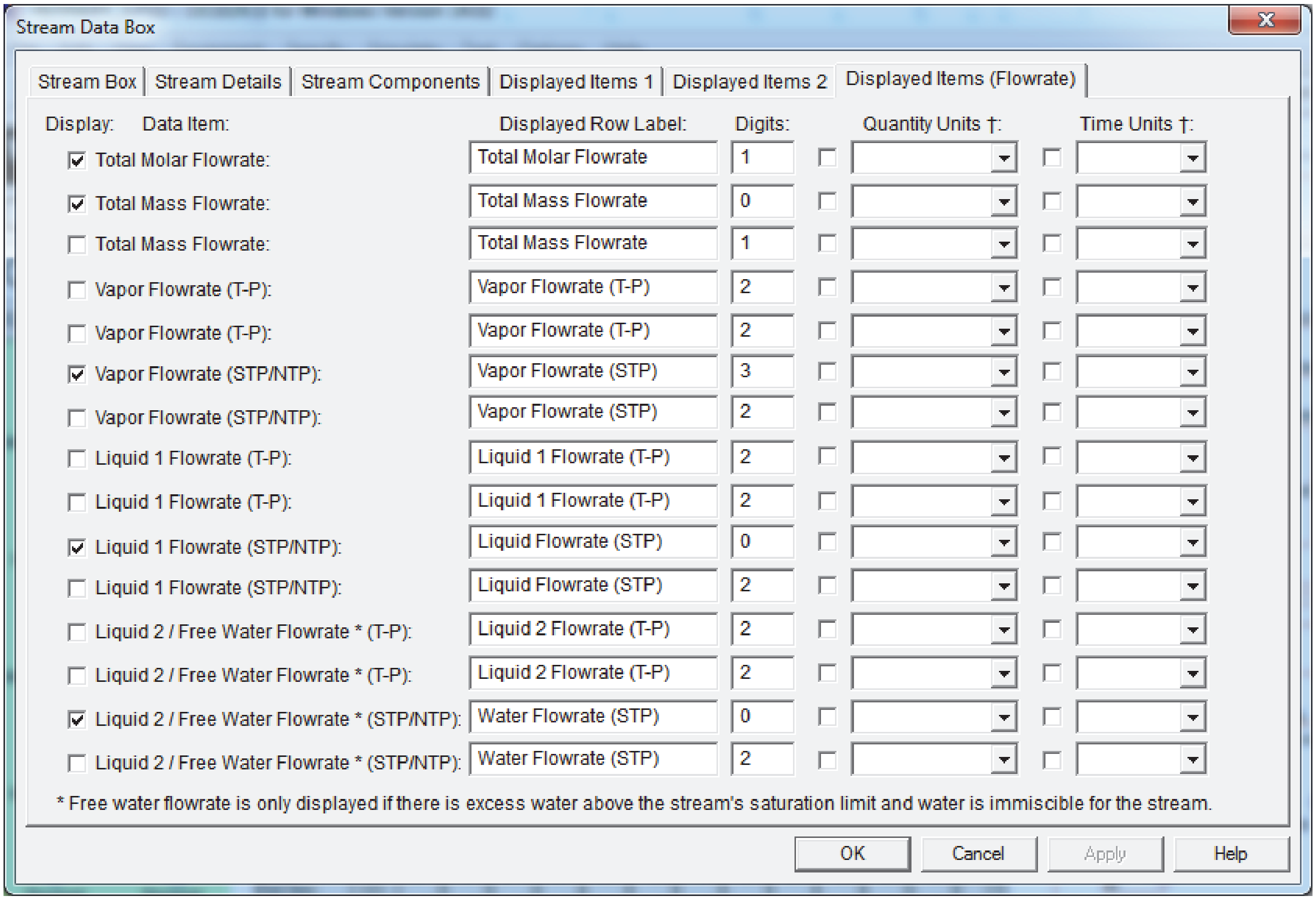


Figure 9: Displayed Items (Flowrate) Tab

You can display the

- Total Molar Flowrate
- Total Mass Flowrate (2 items)
- Vapor Flowrate (T-P) (2 items)
- Vapor Flowrate (STP/NTP) (2 items)
- Liquid 1 Flowrate (T-P) (2 items)
- Liquid 2/Free Water Flowrate * (T-P) (2 items)
- Liquid 2/Free Water Flowrate * (STP/NTP) (2 items)

* Free water flowrate is only displayed if there is excess water above the stream's saturation limit and water is immiscible for the stream

You can select Quantity and/or Time units.

Manage Datasets

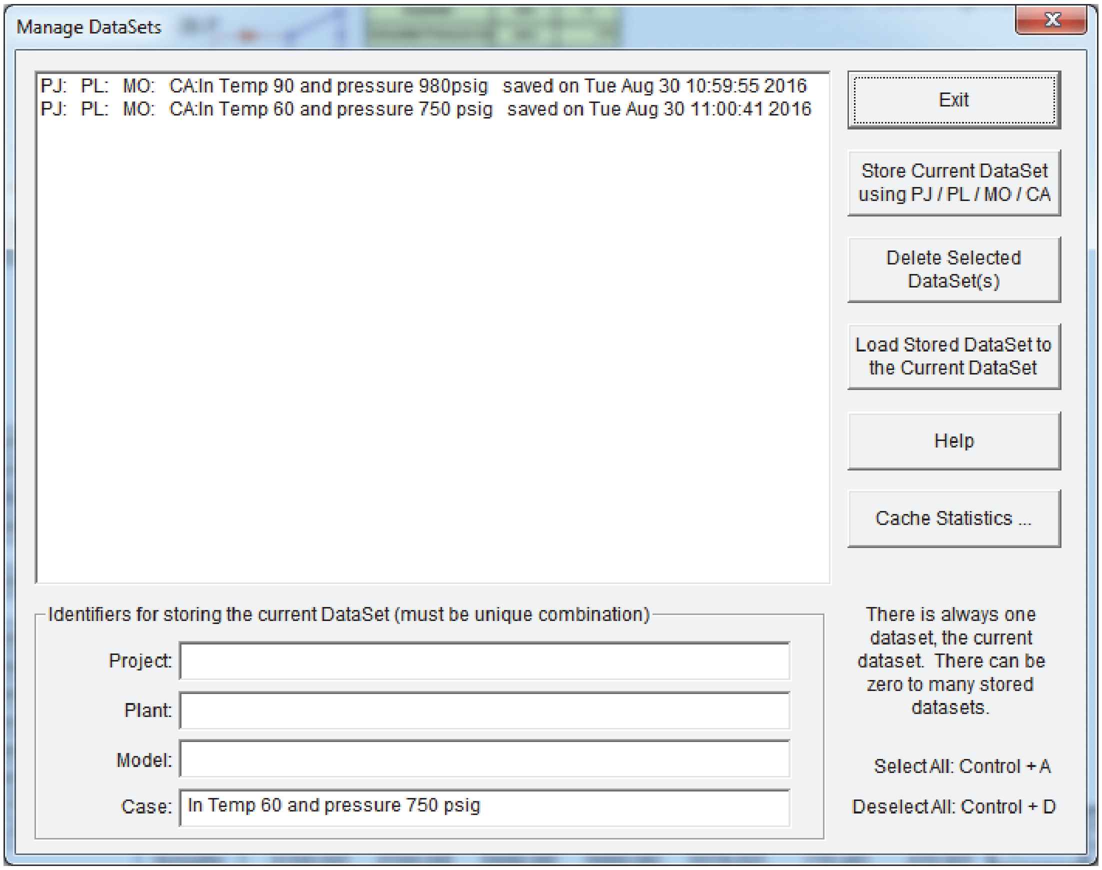


Figure 10: Manage Datasets

This dialog allows you to save a snapshot of your current flowsheet configuration, settings and simulation data providing unique descriptor names (Project/Plant/Model/Case), or to load up a previously saved snapshot.

Also, you can delete saved Datasets here.

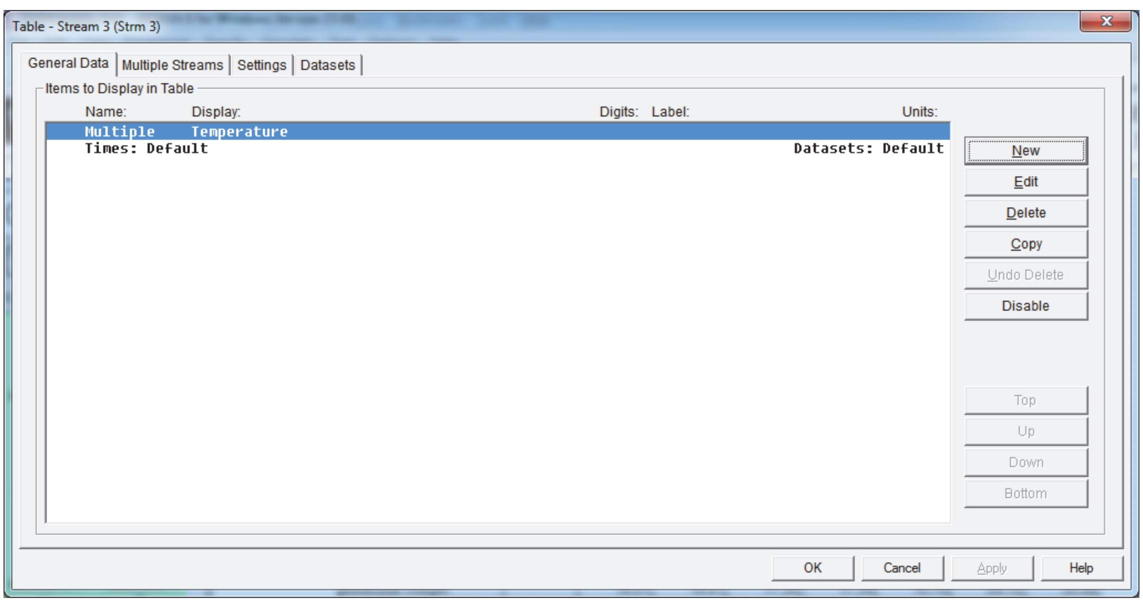


Figure 11: Stream Table Objects: General Data

This dialog allows you to choose several results to display in this table, from both Streams and Equipments. Use the New button to add results, Edit to make changes to existing results, Delete to remove a result, Copy to duplicate an existing result, Undo Delete to bring back the last deleted result, and the Enable/Disable button to turn on/off a result's display in the table.

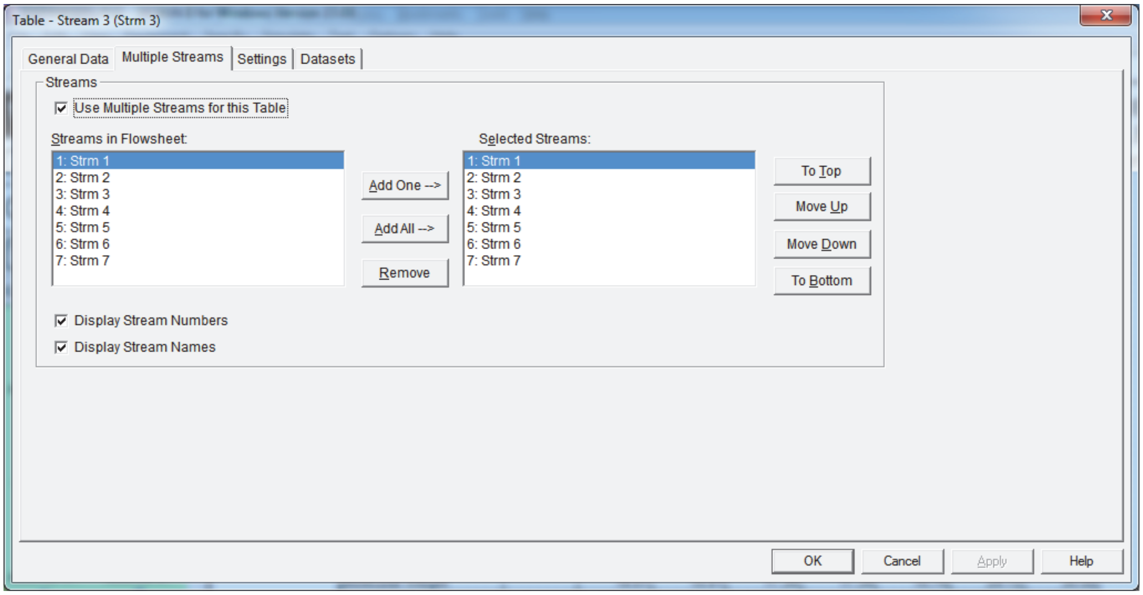


Figure 12: Stream Table Objects: Multiple Streams

This dialog allows you to select which Streams to display for all display items in the current table. Using this feature will no longer allow this table to display results from Equipments.

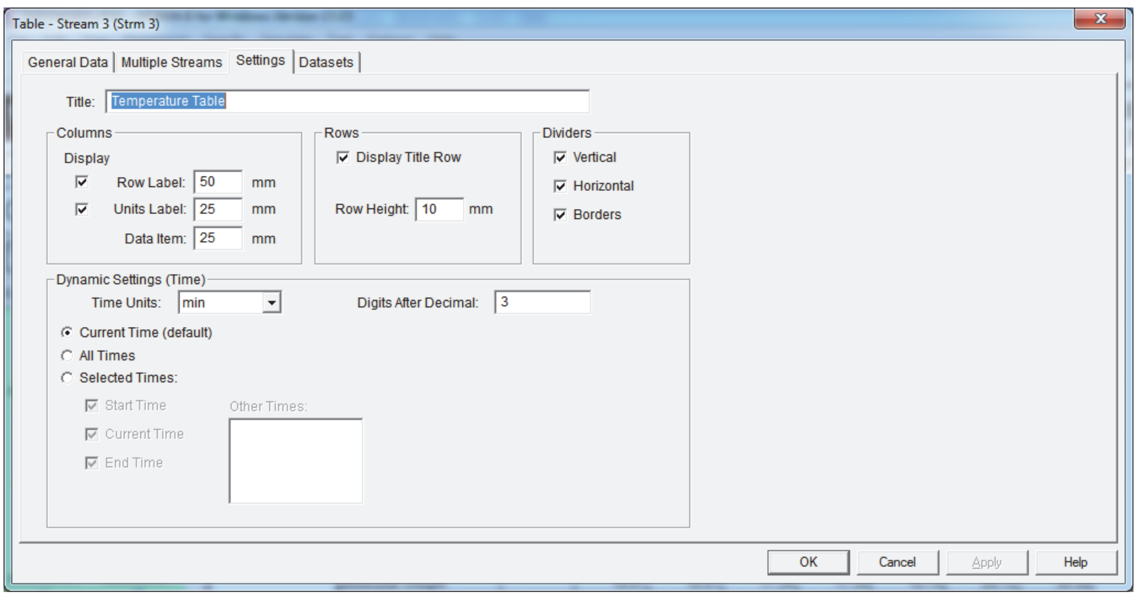


Figure 13: Stream Table Objects: Settings

This dialog allows the user to adjust general display options and sizing for the current table object.

Also, you can choose which time(s) to display for all items displayed in this table that use the "Default" time setting (for Dynamic Simulation Flowsheets.)

Current Time refers to the time step from which the display results are displayed on the flowsheet. The current time can be changed by using the standard playback icons found near the right side of the toolbar at the top of the application. The current time will be shown in the status bar at the bottom of the application, as well as in the table itself.

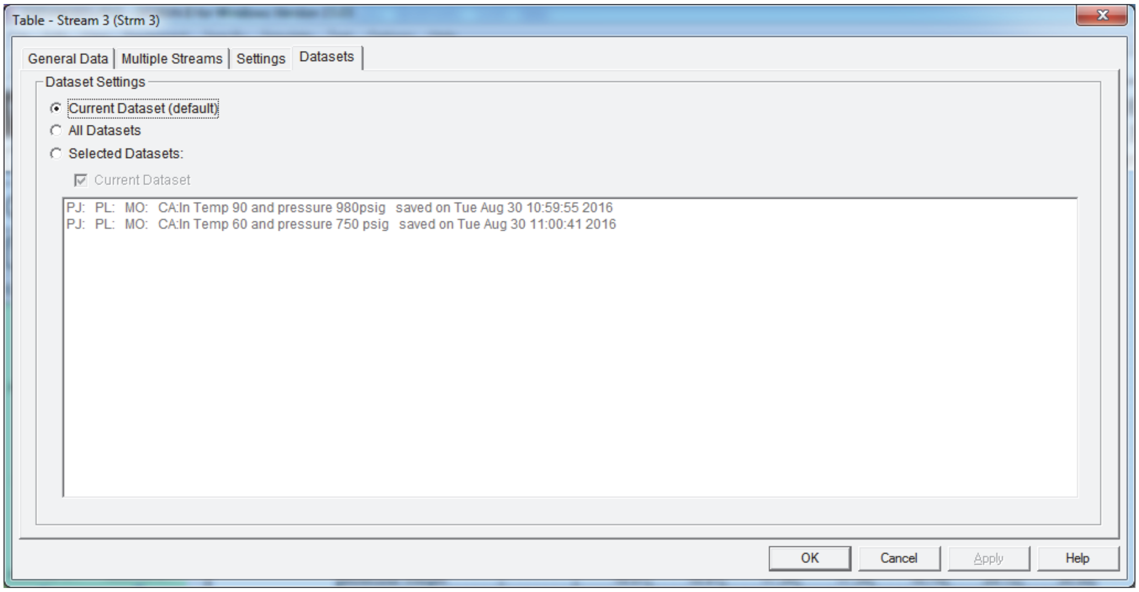


Figure 14: Stream Table Objects: Datasets

The dialog allows the user to select which Datasets to display for each item in the current table that use the "Default" Dataset setting.

The user can choose to select anywhere from one Dataset to all Datasets.

Individual Stream or Equipment

You can view the results on any stream or equipment by double clicking on it and by clicking “View Results” on its Property Dialog.

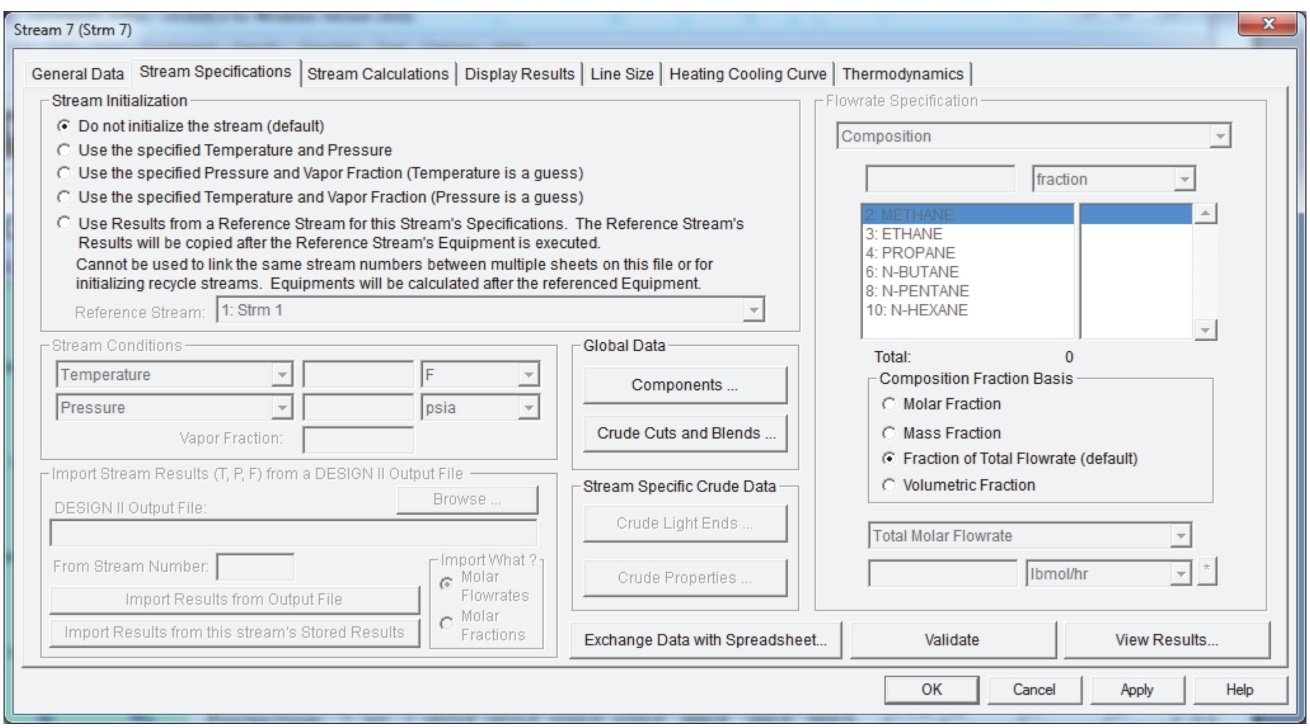
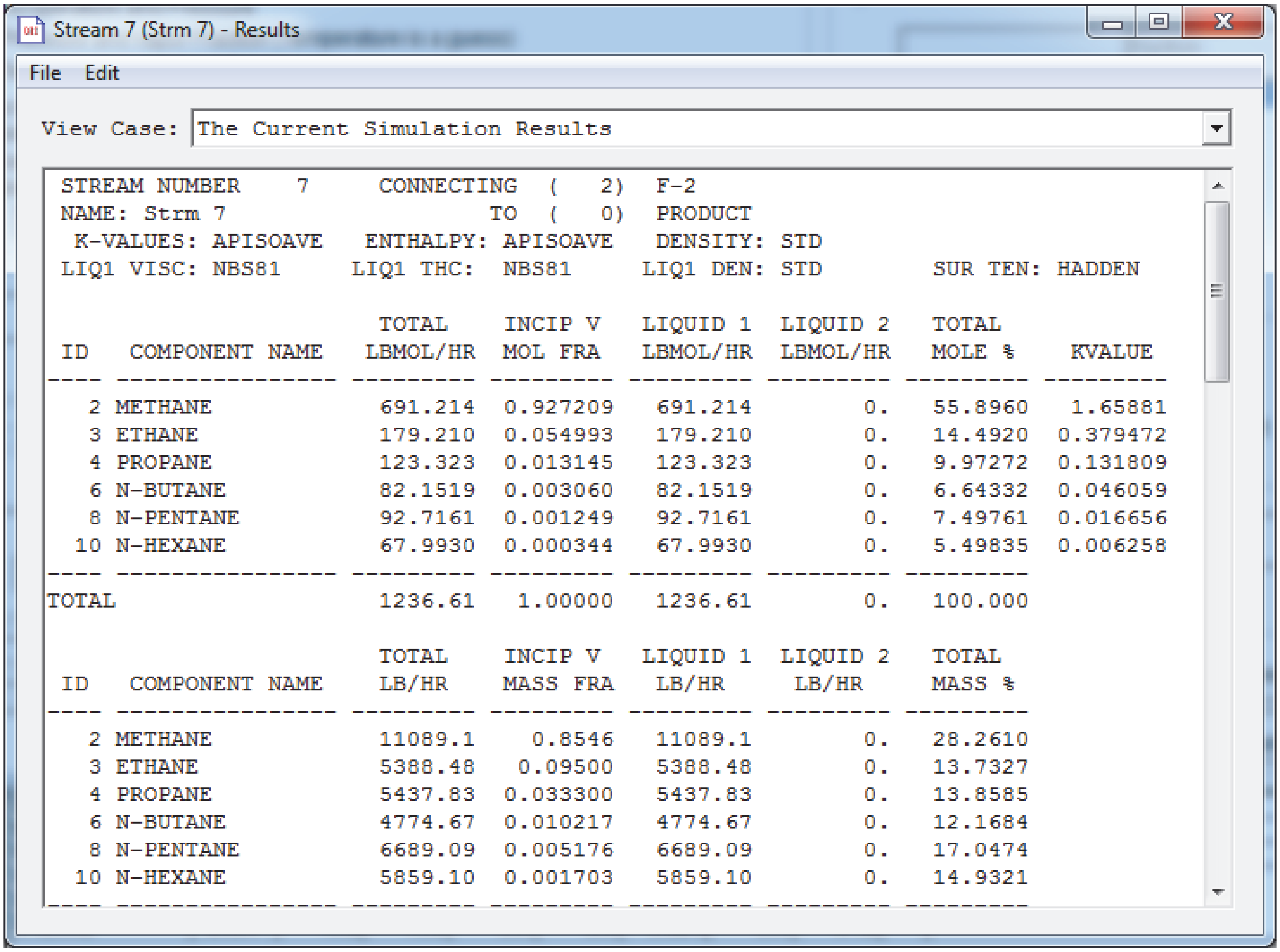


Figure 15: Stream Properties Dialog (from expander.psd)



ID	COMPONENT NAME	TOTAL LBMOL/HR	INCIP V MOL FRA	LIQUID 1 LBMOL/HR	LIQUID 2 LBMOL/HR	TOTAL MOLE %	KVALUE
2	METHANE	691.214	0.927209	691.214	0.	55.8960	1.65881
3	ETHANE	179.210	0.054993	179.210	0.	14.4920	0.379472
4	PROPANE	123.323	0.013145	123.323	0.	9.97272	0.131809
6	N-BUTANE	82.1519	0.003060	82.1519	0.	6.64332	0.046059
8	N-PENTANE	92.7161	0.001249	92.7161	0.	7.49761	0.016656
10	N-HEXANE	67.9930	0.000344	67.9930	0.	5.49835	0.006258
TOTAL		1236.61	1.00000	1236.61	0.	100.000	

ID	COMPONENT NAME	TOTAL LB/HR	INCIP V MASS FRA	LIQUID 1 LB/HR	LIQUID 2 LB/HR	TOTAL MASS %
2	METHANE	11089.1	0.8546	11089.1	0.	28.2610
3	ETHANE	5388.48	0.09500	5388.48	0.	13.7327
4	PROPANE	5437.83	0.033300	5437.83	0.	13.8585
6	N-BUTANE	4774.67	0.010217	4774.67	0.	12.1684
8	N-PENTANE	6689.09	0.005176	6689.09	0.	17.0474
10	N-HEXANE	5859.10	0.001703	5859.10	0.	14.9321

Figure 16: Stream Results Dialog (from expander.psd)

Displaying Stream Results on the Flowsheet

You can view the results (temperature, pressure, vapor fraction, vapor flowrate, liquid flowrate, free water flowrate, enthalpy, total molar flowrate and total mass flowrate) for any stream on the flowsheet. Double click on the stream and select the items that you want to see in the “Display Results on Flowsheet” dialog. If there are simulation results, those items selected will appear on the flowsheet near the stream. Initially, the results will be on top of each other (if more than one selected) and you will need to use the text tool to position them according to your needs. The results on the flowsheet will automatically be updated each time a simulation is run.

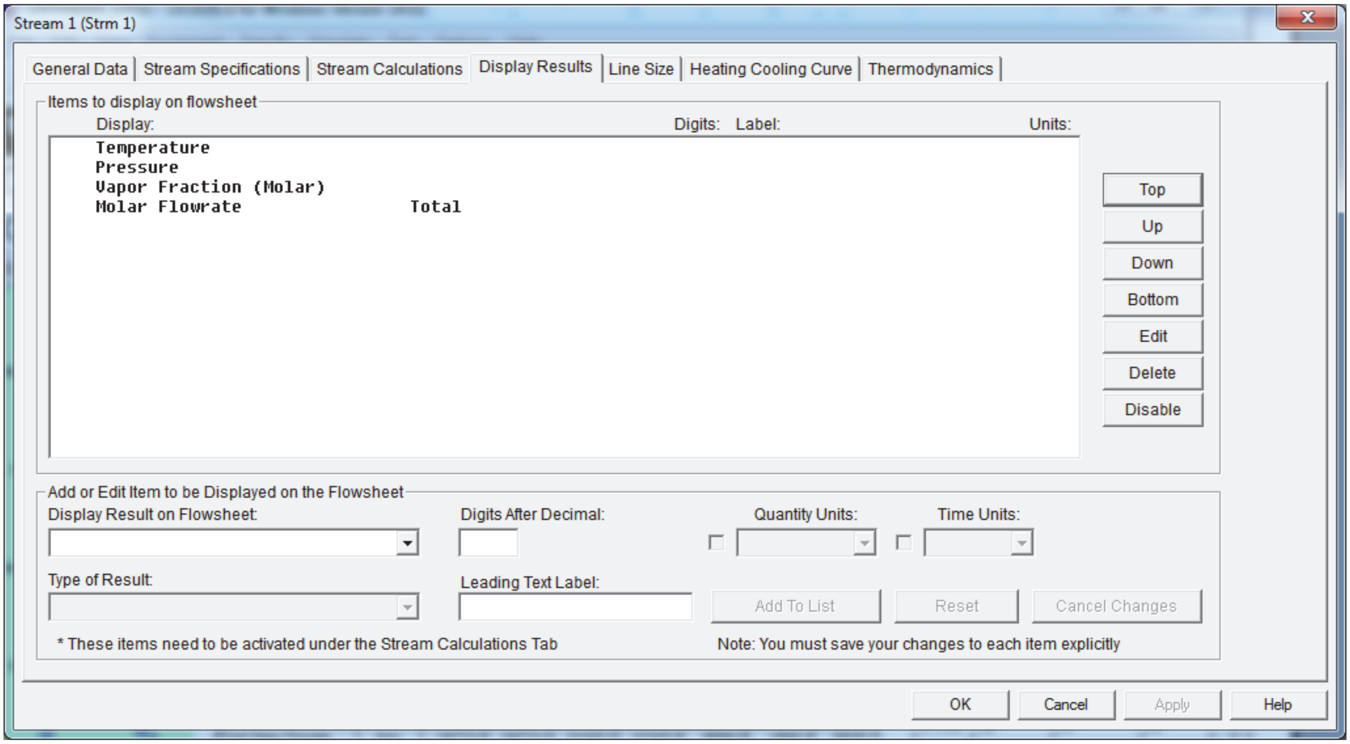


Figure 17: Stream Display Results tab on Stream Dialog (from expander.psd)

Displaying Equipment Results on the Flowsheet

You can view the results (duty, area, Heat transfer coefficient, power, etc…) for most equipment modules on the flowsheet. Double click on the equipment and select the items that you want to see in the "Display Results on Flowsheet" group box. If there are simulation results, those items selected will appear on the flowsheet near the equipment. Initially, the results will be on top of each other (if more than one selected) and you will need to use the text tool to position them according to your needs. The results on the flowsheet will automatically be updated each time a simulation is run.

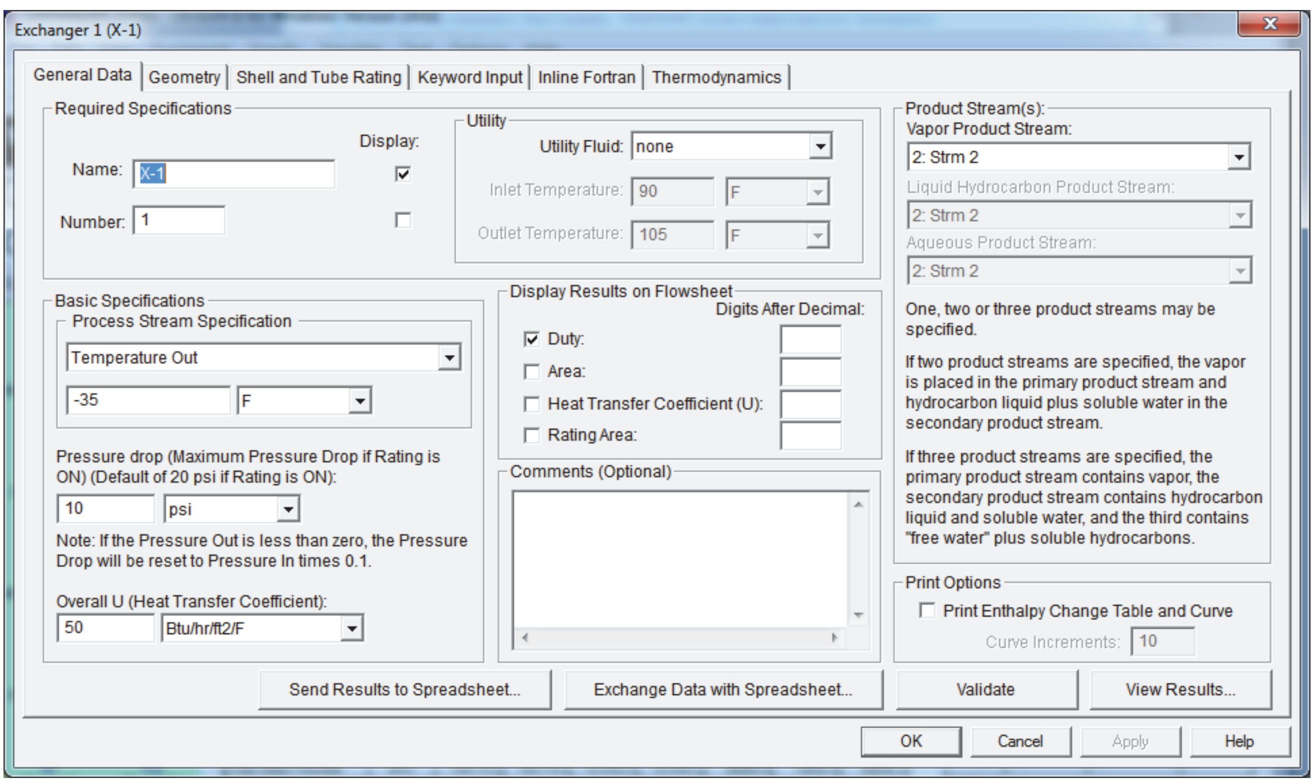


Figure 18: Heat Exchanger Properties Dialog (from expander.psd)

Index