

User's Guide

Release 2.9

Versions 148 onward

Table of Contents

| | |
|---|-----------|
| Introduction..... | 1 |
| On-line Help..... | 1 |
| The User Interface | 1 |
| The Title Bar | 1 |
| The Tool Bar | 1 |
| Configuring the Software | 2 |
| General..... | 2 |
| Interface Language | 2 |
| PFD..... | 3 |
| Solver | 3 |
| Filepath..... | 3 |
| Graphics Attributes..... | 3 |
| Design Grid and Flow Diagram..... | 3 |
| Toolbar | 3 |
| Using Units..... | 3 |
| Using the ‘New Project’ Wizard..... | 4 |
| Previewing Data Files..... | 5 |
| Card Index Editors..... | 5 |
| Report Window..... | 6 |
| Flowsheet Editor | 7 |
| Graphics Display..... | 8 |
| Physical Property Interfaces..... | 8 |
| Using the Built-in Physical Property Engine | 9 |
| Using the Real-time HYSYS/UniSim Interface | 9 |
| Using the Real-time Aspen Properties Interface..... | 9 |
| Importing the Physical Property Source File | 9 |
| The Equation Editor | 10 |
| Data Import and Export Procedures | 12 |
| Using the ColPROF File Converters..... | 12 |
| Installation..... | 12 |
| Using the Aspen+ Converter | 13 |
| Using the HYSYS Converter | 13 |
| Getting technical support..... | 14 |
| How to Get Technical Support | 14 |

Introduction

This manual provides an introduction to the general features of the Process of Integration software suite. More detailed help on specific aspects of each package can be found in the on-line help system.

On-line Help

Help is available from anywhere in the programs by pressing the **F1** key. This will open help at the contents page. Selecting **Help/Contents** from the menu **or** clicking the question mark button on the toolbar will also open help at the contents page. In addition, many dialogues have a **Help** button, which will open the help system at the relevant page. The help system is fully searchable and printable. See the help topic **How to Use Help** (within the help system) for full details.

Some help pages have visual demonstrations that use video and, in some cases, audio to describe how to use a particular aspect of the software. These are indicated by a video camera icon:-



Most of the packages contain technology overview slideshows, to introduce the fundamentals of the technology incorporated within the software. These can be found by clicking on **Technology Overviews** on the help contents page, and are indicated with a slideshow icon:



Some of the packages also include on-line tutorials, which show how to use the supplied example files. Select **Help/Contents** from the main menu, scroll down, and click on **Using the Demonstration Files**.

The User Interface



All of the packages have the same basic user interface:-

The main window contains a title bar, a menu bar, a tool bar for quick access to the most commonly used menu options, and at the bottom of the window, a status bar. The status bar is used to display various program status messages while the software is being used.

The Title Bar

The title bar shows the path and filename of the currently loaded file. If the filename is followed by **<R>**, it is read-only and can only be saved with a different name. If the filename is followed by **<M>** it has been modified since it was last saved. Double clicking the title bar will maximise the window.

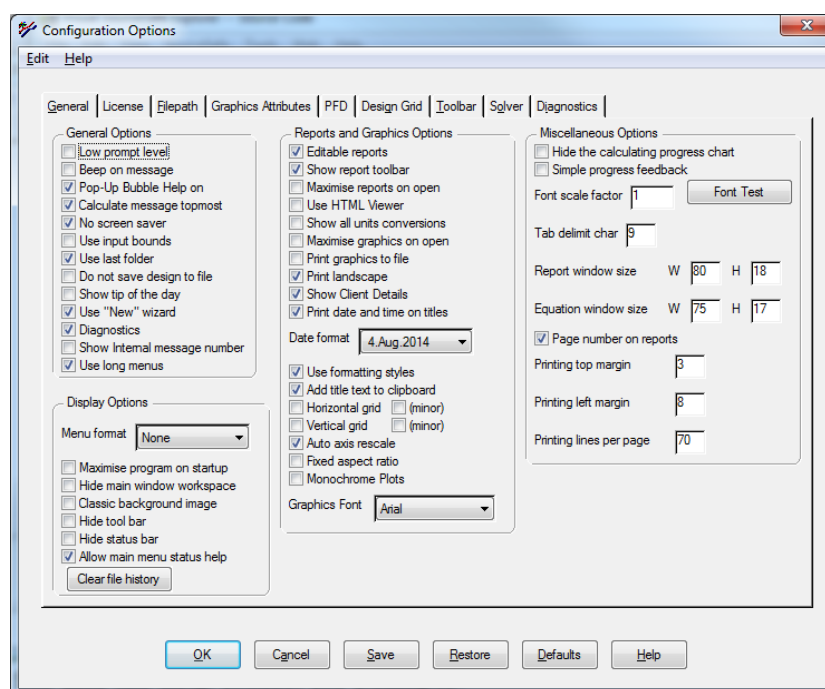
The Tool Bar

The tool bar provides quick access to some of the more common menu items. If the **Popup Bubble Help** option is active in the program configuration, holding the mouse pointer over one of the tool bar

buttons for a short moment will display a help message indicating its function. The tool bar can be customised to include only the buttons used most frequently.

Configuring the Software

Many aspects of the programs may be configured, to suit your personal preferences. Start any of the programs and select the **File/Preferences** menu to open the configuration dialogue: -



This dialogue is in the form of tabbed property sheets. Some or all of the following property sheets will be available, depending on which program you are running. A full description of each of these sheets is available in the on-line help.

General

This contains general user preferences, mostly relating to the user interface and the appearance of the graphical output windows. Place a check mark next to any of the options to select it.

Interface Language

Some applications have an option to change to interface language. If the language is changed from the default (English (UK)), it will try and read the appropriate language dependent files stores in the `CPI_Suite\<application>\Lex_data` folder. If a word or phrase cannot be located in the language file the internal English version will be used. The format of the put dialogues are based on the number of characters the English words so non-English dialogues may have some layout issues. To maintain accelerator key functionality, if the non-English characters string does not have the appropriate character then the accelerator appended to the end of the string (e.g. `Save file - > Enregistre le fichier(v)`). When using a language that is not based on the western alphabet (i.e. Chinese, Arabic etc.) then reports printing may become an issue. In these cases, it is advised that the output of the application is copied into a more language friendly package (e.g. Microsoft word) for printing

If you are using a non-western language setting you may need to adjust your window language setting to use

WIN 10 : - Install the preferred language (if necessary)

Start\windows settings\time and language\language\Add a preferred language

As the applications are non-unicode, you may need to adjust the setting

Administrative language settings\administrative\change system locale

To the appropriate language

Win 7: - Install the preferred language (if necessary)

Start\Control panel\region and language\keyboards and languages\Install/uninstall languages

As the applications are non-unicode, you may need to adjust the setting

Start\Control panel\region and language\administrative\change system locale

To the appropriate language

PFD

Contains options relating to the Process Flowsheet Diagram

Solver

Some of the packages allow you to choose GAMS as your primary solver, rather than the built in solvers.

Filepath

This allows you to set the location of various auxiliary files and folders used by the software. These include your security key file and the temporary folder, for example.

Graphics Attributes

This allows you to customise the appearance of flowsheets, by modifying line colours and styles.

Design Grid and Flow Diagram

These property sheets allow you to select which information is displayed on design grids and flow diagrams and to sort process streams by several different criteria.

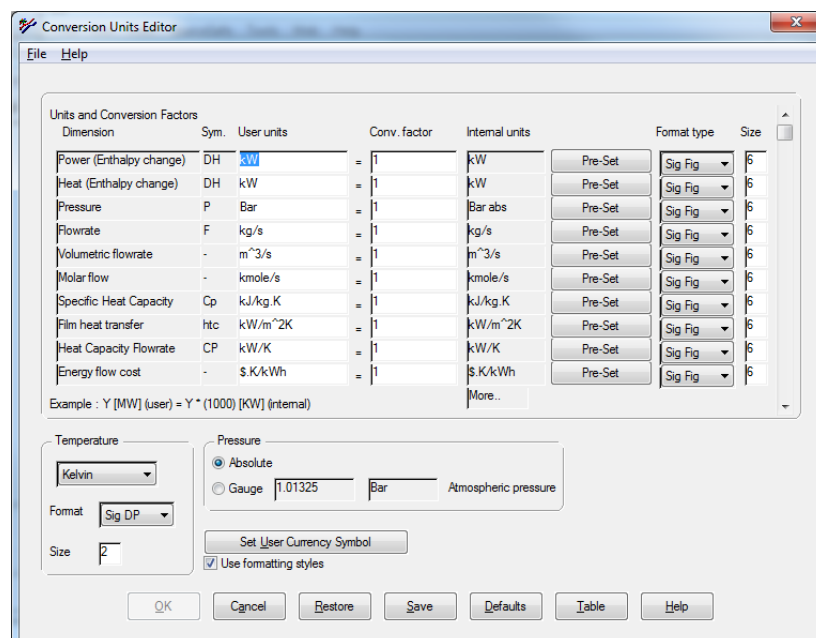
Toolbar

Allows you to customise the toolbar by including buttons for the programs features you use most frequently.

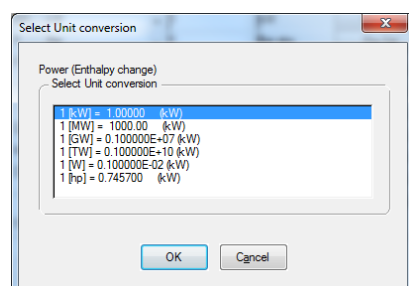
Click the **Save** button to save the changes to the configuration. The **Restore** button will restore the settings to the previously saved configuration, and the **Defaults** button will restore the configuration to the program's internal defaults.

Using Units

The programs have been designed to allow users to specify their own dimensional units. All units are selected from the **Units Editor**, which is accessed from the **Edit** menu.



Each unit has a text description, and a conversion factor to convert the user-defined units into the program internal units. For example, to change the Enthalpy units to megawatts, enter “MW” in the user units box, and 1000 in the conversion factor box. (1MW = 1000kW). For each unit class there are some “Pre-Set” values that you exploit to build your preferred units conversion set.



Whenever a value has a unit associated with it, the currently selected unit is displayed in square brackets – [kJ/kmol]. This convention is used for user input, and for reports. In some instances, where correlations are involved, a value must be entered in a specific unit. In this case the unit is displayed in round brackets – (dynes/cm).

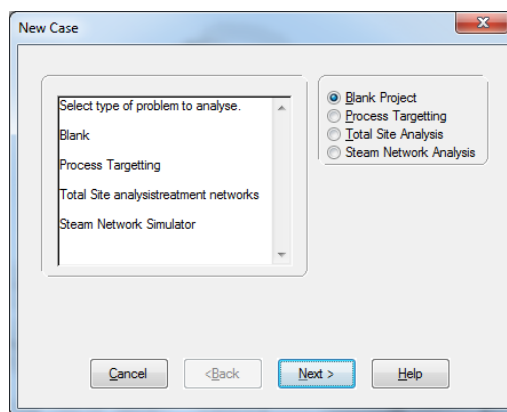
To export the current unit set to a different project file:

- With the first project file open, select **File/Export Unit Set**, and select a file name for the unit set
- Open the second project profile, open the Units Editor (**Edit/Units**) and select File/Import Unit Set

This procedure may be carried out even for project files in different CPI applications. For example the units set in a COLSEQ file can be exported to a project file in STAR.

Using the ‘New Project’ Wizard

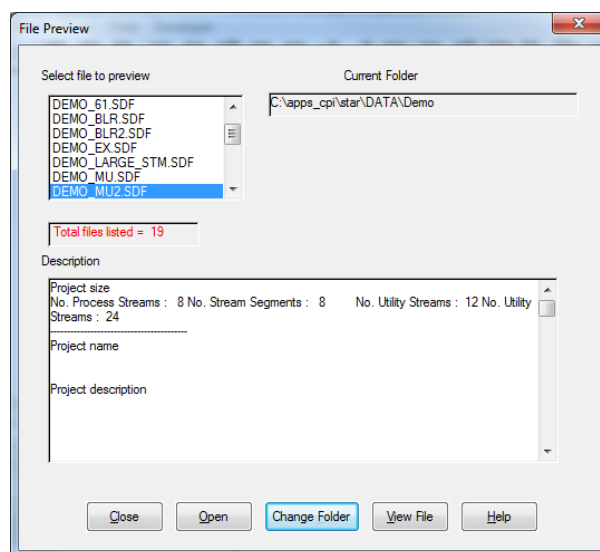
Most of the packages have a **New Project** wizard to guide you through the basic steps involved in setting up a problem. Make sure the **Use “New” Wizard** is selected in the configuration options, then select **File/New**, or click the **New Project** toolbar button.



For each data item, click **Edit** to enter the editor. Click **Next** to advance through the editors, or **Back** to move backwards. The wizard can be closed at any time by clicking the Finish button.

Previewing Data Files

It is possible to preview the contents of data files, without having to open the file. Select **File/Preview**, to open the File Preview editor: -



The preview box contains a list of data files in the current directory. Clicking on a file in the list will display a summary of the contents of the file, and the file description (the description of any file can be modified in the **Project Description** editor – select **Project Description** from the **Edit** menu)

To open a file directly from the Preview Editor, either select its name and click **Open**, or double-click the file name. The complete contents of the selected file can be viewed by clicking **View**.

To change directories click **Change Dir...** and browse for any file in the required directory.

Card Index Editors

Most of the packages make use of card index editors, for adding items such as feeds, products or utilities.

When these editors are first opened, all the entry boxes will appear greyed out. Clicking the **Insert** button will add an item, and the various entry boxes will become available, to enable the properties of the new item to be defined. When multiple items are present, clicking the spin-box at the top of the window (or pressing the **Page Up** or **Page Down** key) will display the next item in sequence. The **Delete** button will delete the currently displayed item, and the **Table** button will display a report of all items, for printing or pasting into a report.

Most of these editors contain an **Edit** menu, with options for copying and pasting new items.

Most of these editors have access to the standard windows Calculator (F11) and the standard windows Notepad (Shift+F11) to permit quick access.

The report viewer can be toggled (Ctrl+F11) between the standard internal report windows and a HTML based viewer with enhanced functionality.

Report Window

The text output from each program is displayed in a standard resizable report window.

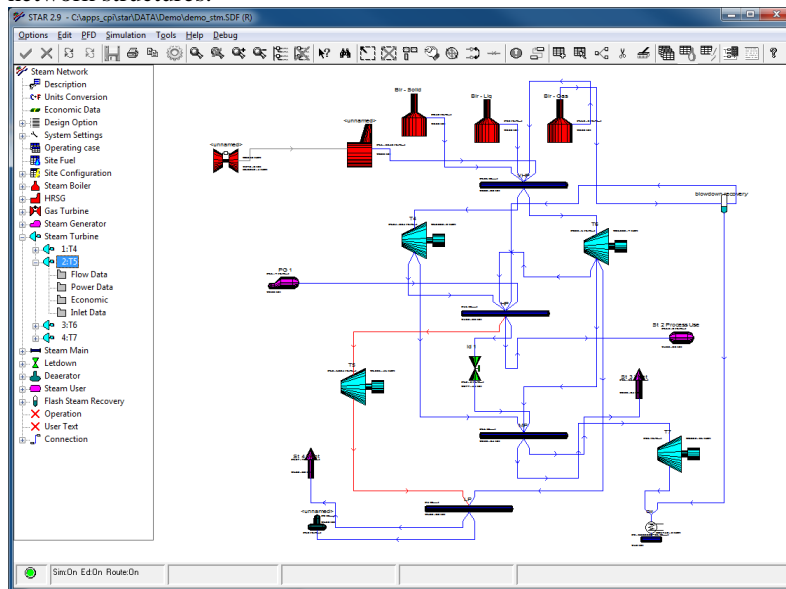
The contents of the report can be printed by clicking the Print button. The **Options** menu contains functions to save the report as a text file, to copy it the Windows clipboard, to find text in the report and to jump to a specified line number.

If the **Editable Reports** option is checked in the main configuration window, the report can be edited before printing, for example to modify the title, or to remove redundant information.


If the “HTML Viewer” is selected in the preferences then alternative viewer is displayed (currently the default windows web browser but is intended to be replaced with an application based viewer) which (if available) will have links to various sections with the reports.

Flowsheet Editor

Most of the packages make use of the drag-and-drop flowsheet viewer/editor for viewing or creating network structures.



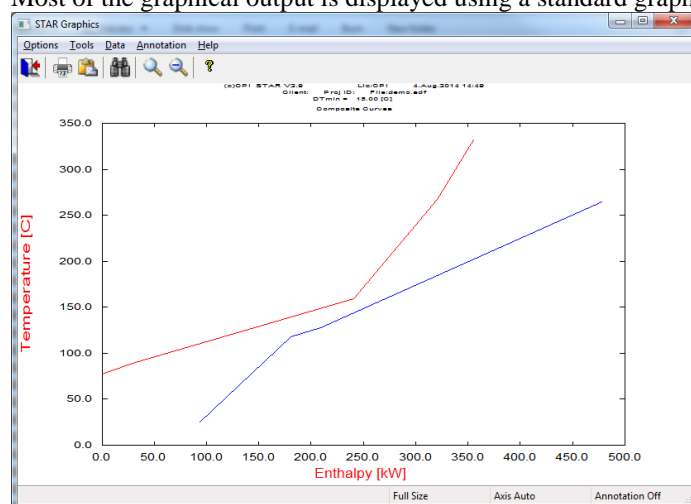
The flowsheet viewer has its own tool bar with various functions for manipulating the image, or adding and editing units and connections. Moving the mouse pointer over a toolbar button will indicate its function in the status bar at the bottom of the window. For example, to move a unit:-

- Click the move button . The mouse pointer will change shape to indicate move mode.
- Move units by dragging them to the required location.
- Exit from move mode by right-clicking the mouse.

The exact functionality of the flowsheet editor and the available buttons will depend on the program being used – see the on-line help for details.

Graphics Display

Most of the graphical output is displayed using a standard graphics display window: -



The graphics window menu bar has the following items:-

- **Exit.** Closes the graphics window, and returns control to the program
- **Print.** Prints the display (as it currently appears) to the current default windows printer
- **Zoom.** Drag a rectangle to zoom in on any area of the display
- **Full** Zooms out to display the complete graph
- **Locate** Displays a sub-menu to either display the x-y co-ordinates of the cursor (**Coords**), or to supply additional context-sensitive information if available (**Identify**)
- **Axes** Allows the axes scales to be modified. The display graph titles can also be modified.
- **Marks** Toggles marker symbols at each x-y location
- **Attrib.** Allows various attributes of the graph, such as line co-ordinates, to be displayed in a report. In the case of multiple graphs being plotted on the same axes, it also allows individual graphs to be switched on or off..
- **Annot.** Allows annotations and legends to be added to the display. To add an annotation select **Annot./Add**, type the annotation, press return, then click on the display where you would like the annotation to appear. To add a legend, select **Annot./Legend**, and drag a rectangle where you would like the legend to appear.
- **Help** Opens the help system at the relevant page

Note that while the graphics window is open, none of the menu items or toolbar buttons will be available. The graphics window must be closed before control returns to the program.

A message in the status bar indicates whenever the graphics window is open.

By default if any data is modified, when the application return to the main window the data is automatically saved (~\$<filename>). This can be used to recover project data in the event of a failure. By default, these backup file are automatically deleted on normal termination of the application

Physical Property Interfaces

Several of the packages make extensive use of physical property calculations. Physical property calculations can currently be performed in a number of different ways. Each method involves creating a **Physical Property Source** file, which is then loaded into the software in the same way, as described below.

Using the Built-in Physical Property Engine

To use the built-in physical property engine it is necessary to create a data file containing all the parameters necessary to calculate the properties. These files are called **Component Data Files** and have a **.CMP** extension. Component Data Files may be created using the **Component Editor** program that is supplied as part of the software suite. Alternatively, most commercial simulators include scripting languages which will allow properties to be exported to a file. Contact us for details of the Component Data File format if you would like to take advantage of this feature.

When using a Component Data File, all physical properties will then be calculated using the program's own internal routines. Full details on how to create a Component Data File can be found in Component Editor's on-line help.

Using the Real-time HYSYS/UniSim Interface

If you are licensed users of Aspen Technology's HYSYS or Honeywell's UniSim process simulator, it is possible to make use of its physical property engine and component database. The procedure is as follows: -

- Start HYSYS/UniSim, and create a new fluid package, consisting of your required components and property package.
- Switch to the simulation environment and create an empty stream, named **DPL_Temporary_Stream**.
- Save the file (with an HSC/USC extension) and then exit from HYSYS/UniSim.
- Import the HSC/USC file into the required Process Integration application, as described below.

When this option is chosen all physical properties are calculated by HYSYS/UNISIM. If the **Show HYSYS/UNISIM User Interface** option is checked, on the main configuration window, HYSYS/UNISIM will open as a normal window, allowing the current stream properties to be examined. If this option is not selected, HYSYS/UNISIM will open and run invisibly in the background.

Using the Real-time Aspen Properties Interface

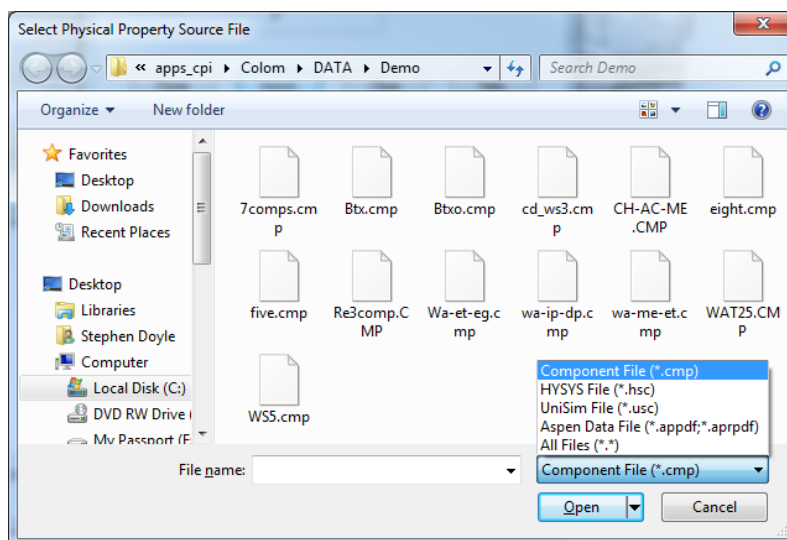
If you are licensed users of the Aspen Properties process simulator, provided by Aspen Technology, you can use its thermo server for physical properties.

Before using the Aspen thermo server you must configure your computer so that the CPI software can find the necessary Aspen files. The procedure is fully documented in the on-line help system.

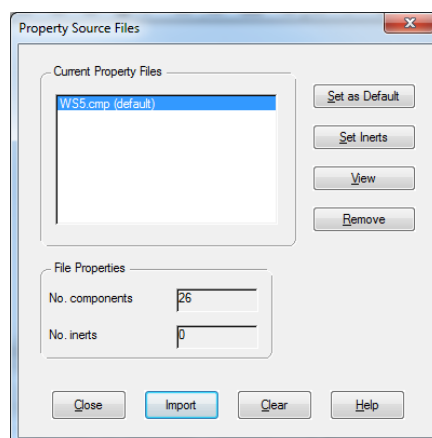
From any of the packages select **Help -> Contents**, and then select **General -> Physical Property Data->Using Aspen**.

Importing the Physical Property Source File

Once you have created your physical property source file containing the required components and property package, it must be imported into the respective package. The method will depend on the package you are using. In COLSEQ select **Edit/ Physical Properties Data Source**, then click the **OK** button. This will open a standard File Open dialogue:-



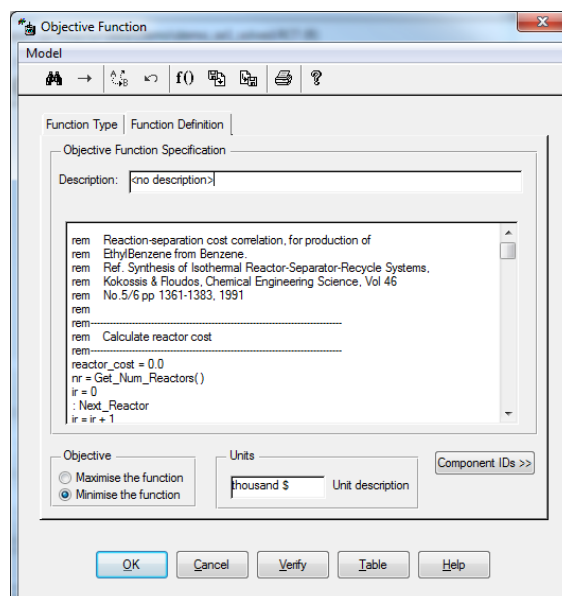
In the File Type drop-down box select the required file type, **Component file (CMP)**, **HYSYS file (HSC)**, **UniSim file (USC)** or **Aspen Data file (appdf, aprpdf)** and then browse for and open your file. This will open the Physical Property Source dialogue:-



Click the **Open** button to open a new file, the **Summary** button to view summary information for the file, or the **Clear** button to clear the existing file.

The Equation Editor

The **Equation Editor** is a feature that enables users to supply their own correlations, for such things as objective functions, reaction kinetics, and refinery models. The editor works by supplying a list of interface functions to the program's internal variables, and allowing the results of calculations to be returned back to the program. The basic features of the Equation Editor are the same in each package, although the available interface functions vary. The following example shows how the equation editor can be used to calculate the cost of a reactor network in the REACTOR program.



total_reactor_cost = 0.0

Initialises the total reactor network cost to 0.0.

nr = Get_Num_Reactors()

This is an example of a function call. A function call is followed by round brackets, which may contain a list of arguments. This call sets nr to be equal to the number of reactors in the network

ir = 0

Sets ir to 0

:Next_Reactor

This is an example of a label, as indicated by the colon (:) at the start of the line. Labels have no effect, but act as locations that can be jumped to with the goto command.

ir = ir + 1

Increments ir, which is now equal to 1.

active = Get_Reactor_Active(ir)

Another function call. This returns TRUE if reactor number ir is active in the network, FALSE otherwise.

if (active = FALSE) goto End_This_Reactor

This is an example of a conditional statement. If active is equal to false, then the program will jump to the label indicated by End_This_Reactor.

rvol = Get_Reactor_Volume(ir)

Another example of a function, this time returning the volume of reactor number ir.

reactor_cost = 100 + (rvol * 10)

Calculates the cost of this reactor.

total_reactor_cost = total_reactor_cost + reactor_cost

Adds the cost of this reactor to the running total.

:End_This_Reactor

Another label, indicating where the program should jump to for inactive reactors.

if (ir < nr) goto Next_Reactor

Another example of a condition. This tests to see if the current reactor number is less than the total number of reactors, and if so loops back to consider the next reactor.

return total_reactor_cost

This returns the total reactor cost network to the program. In some of the packages, values can also be returned using function calls.

The scripting language used is a subset of the BASIC programming language, although line numbers are not required. Full details of the command syntax used, and the available function calls are described in each package's on-line help, but note the following general points:-

- The language is completely case-insensitive. Mixed upper and lower case may be used to increase readability.
- It is not necessary to create the text in the equation editor window. Any text editor can be used to edit and save the text, which can then be imported using the **Edit/Import from File** option on the equation editor window. Imported files have the default .EQN extension.
- A list of the available functions and parameters can be viewed with the **Edit/Insert Function** and **Edit/Insert Parameter** menu options. Selecting a function or parameter and then clicking the OK button will insert the function text in the editor window at the current cursor location.
- **Find, Replace, Goto Line** and **Undo** facilities are also available under the **Edit** menu.
- Functions can be used to return values back to the program, as well as to interrogate program variables. In general functions which interrogate the program begin with **Get_** and functions which return values begin with **Put_**.

Data Import and Export Procedures

Some of the packages allow data to be transferred between them using export and import functions.

To export targeting data from **ColSEQ** to **STAR** or **SPRINT**:-

- In the **ColSEQ Target** menu. Either **Task-based Sequencing** or **SA/GA Sequencing** must have been performed
- Carry out the calculations using the normal procedure, then select **Target/Export Stream Data File** to export an SDF file.
- To import into **STAR**, select **File/Open**, and open the file created in step 2.
- To import into **SPRINT**, select **File/Import**, select **Star Files (*.SDF)** from the file type dropdown, and open the file created in step 2.

To export stream and utility data from **STAR** to **SPRINT**

- In **STAR**, select **File/Save**, and save an SDF file
- In **SPRINT**, select **File/Import**, select **Star Files (*.SDF)** from the file type dropdown, and open the file created in step 1.

Using the ColPROF File Converters

Two file converter programs are available which allow converged simulations in HYSYS or Aspen+ (Release 9) to be exported directly to **ColPROF** (.CPF) files. These converter programs are not supplied as part of the standard CPI software package but are available, on request, to licensed users of the respective simulator.

Neither converter is a stand-alone program. They require both **ColPROF** and the respective simulator to have been installed. In addition, the Aspen converter requires the Aspen Toolkit to have been installed.

Installation

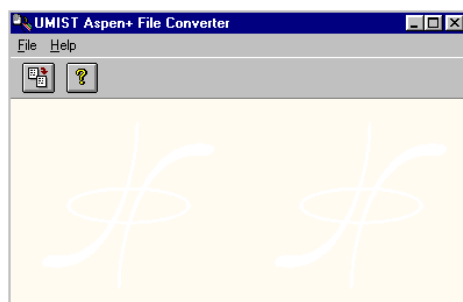
The converters are supplied as a single file self-extracting installation.

To install from the single file simply run the executable (SETUPEX.EXE).

When selecting the installation directory, it is important to choose the directory into which **ColPROF** was installed. By default this will be **C:\Program Files (x86)\CPI Suite**

Using the Aspen+ Converter

If the converter has been installed to the default location (i.e. the same directory as the main software package) the converter can be launched from **ColPROF** by selecting **Edit/Launch Aspen+ Converter**.



From the converter **File** menu, select **Convert**, and browse for your Aspen backup (.bkp) or summary (.sum) file. Once the file has loaded a dialogue will appear, with a list of the columns in the simulation.

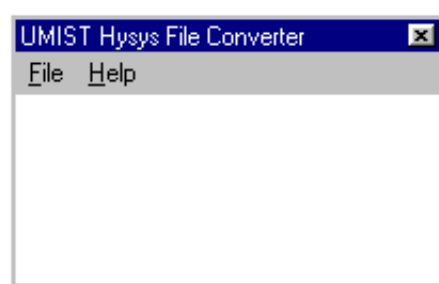
Select the required column (RADFRAC columns only), and click OK. A **Save** dialogue will appear asking for a name for the .col file. Select a filename, click OK, and then load the .COL file into **ColPROF** in the usual way. Column exergy and enthalpy profiles can now be plotted using options under the **Profiles** menu.

More detailed help, describing the properties required in the Aspen simulation, are provided in the File Converter's on-line help. In particular please read the section **Summary File Property Requirements** which describes which user-specified physical properties units must be present in the Aspen simulation.

Please note that the Aspen+ Toolkit must be loaded for the file converter to work. The converter makes use of several files that are only installed as part of the Toolkit. If the file converter gives an error message that refers to missing DLL files or missing DAT files, the Toolkit is not correctly installed.

Using the HYSYS Converter

If the converter has been installed to the default location (i.e. the same directory as the main software package) the converter can be launched from **ColPROF** by selecting **Edit/Launch HYSYS Converter**.



From the converter **File** menu, select **Convert**, and browse for your HYSYS simulation file (.hsc). HYSYS will automatically be launched and the simulation data extracted. If the simulation consists of a single column, a dialogue will appear confirming the column name, otherwise a selection box will appear with a list of the columns in the simulation. Select the required column and click OK.

Finally, a **Save** dialogue will appear asking for a name for the .col file. Select a filename, click OK, and then load the .COL file into **ColPROF** in the usual way. Column exergy and enthalpy profiles can now be plotted using options under the **Profiles** menu.

Getting technical support

How to Get Technical Support

There can be many and varied reasons for requiring assistance when using a program. It is natural, especially for novice users. Please do not think that any question is too trivial other may have the same question and it may help us improve the software or help system for others

What we would expect is that you have taken reasonable steps to resolve the issue yourself. The support question can fall into two broad categories

- **Technology related issues** - These relate to how case studies are approached, what calculations are possible and any workarounds that may be required to handle a situation not directly covered in the application e.g. How to model a Heat pump in energy targeting?. **NOTE:** For University of Manchester students your supervisor should be the primary point of contact for technology based questions. The software support team have explicit instructions not to tell students "How to solve their coursework". They will give guidance on how to use the software.
- **Software related issues** - These usually relate to unexpected program behaviour (i.e. crash) or concerns over a given result, licence file issues etc. **NOTE:** Previous versions of the application are not supported. Please ensure that you have the latest version installed. See Help\About for version and build date information

There is a significant amount of information in both categories in :-

- **The on-line help** - The on-line help (F1) for most applications is the primary source of the description of the application, effect of selecting various options, workarounds etc. The on-line help has a powerful search facility to assist in finding help pages that may be of benefit. The on-line help also has an [FAQ's](#) section covering some common issues that users experience.
- **The support Web pages** - The pages (accessible from the main help menu) contain details of the latest version of the application, downloadable installation images and details of any enhancements or major bug fixes.
<https://www.ceas.manchester.ac.uk/cpi/research/resources/software/>
- **Installation Guide** - This can be useful in identifying where the components of an application are stored and how to configure the system for non-standard installations.

When contacting the support team always:-

- Send a copy of the data file if applicable (data confidentiality would apply to any external based user request)
- What application, version number, build date (see Help\About). Version of windows (XP,8,10 32 or 64bit)
- For unexpected program behaviour the exact sequence of events that can be used to demonstrate the behaviour
- What steps already taken to resolve the issue and their outcome
- Contact details if not a registered student. **NOTE:** Email addresses (i.e. private email etc.) from unknown organisations will not be responded to.

Contact information

The Software Team
Centre Process Integration

Department of Chemical Engineering
5th Floor, Core 4E, MECD Eng Bld A

The University of Manchester
Oxford Road
Manchester
M13 9PL
UK

Tel : +44(0)161 529 3801

Mail : CPITechSup@manchester.ac.uk

Web : www.manchester.ac.uk/cpi

Windows is a registered trademark of Microsoft corporation, <http://www.microsoft.com>

GAMS is available from GAMS Development corporation, <http://www.gams.com>

Aspen Plus and Hysys are © Aspen Technology Inc. <http://www.aspentech.com>

UniSim © Honeywell Process Solutions Inc. <http://www.honeywellprocess.com>