UniSim® Crossflow Exchanger Modeler

(referred to as UniSim® CFE)

Getting Started Guide

Honeywell

Copyright

March 2008 Release R380

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

For further information please contact

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

Prepared in Canada.

Table of Contents

Introducing UniSim® CFE

Introduction

There are two main sources of documentation for UniSim® CFE that will be discussed in the following sections:

- Manuals
- Online Help

Manuals

This Getting Started Guide forms part of the documentation supplied with each UniSim® Heat Exchangers program:

- **UniSim® Heat Exchangers User Guide**
- **UniSim® CFE Getting Started**
- **UniSim® CFE Reference Guide**.

The UniSim® Heat Exchangers User Guide is supplied in pdf format with the installation media. The Documentation media contains the UniSim® Heat Exchangers User Guide and all the other documents. The UniSim® Heat Exchangers User Guide is generic to all UniSim® Heat Exchangers programs. The Getting Started and Reference Guide are specific to each UniSim® Heat Exchangers program.

The Getting Started Guide assumes you have access to an installed copy of the UniSim® Heat Exchangers program, and takes you through some example cases provided with the program, so you can get a feel for its capabilities. It also describes how you can run a set of QA sample cases, and compare the output files with sample results, to confirm that the operation of the program is as it should be.

More detailed examples, showing how you can use the program to solve typical problems, are provided in Appendices to the program Reference Guide.

Online Help

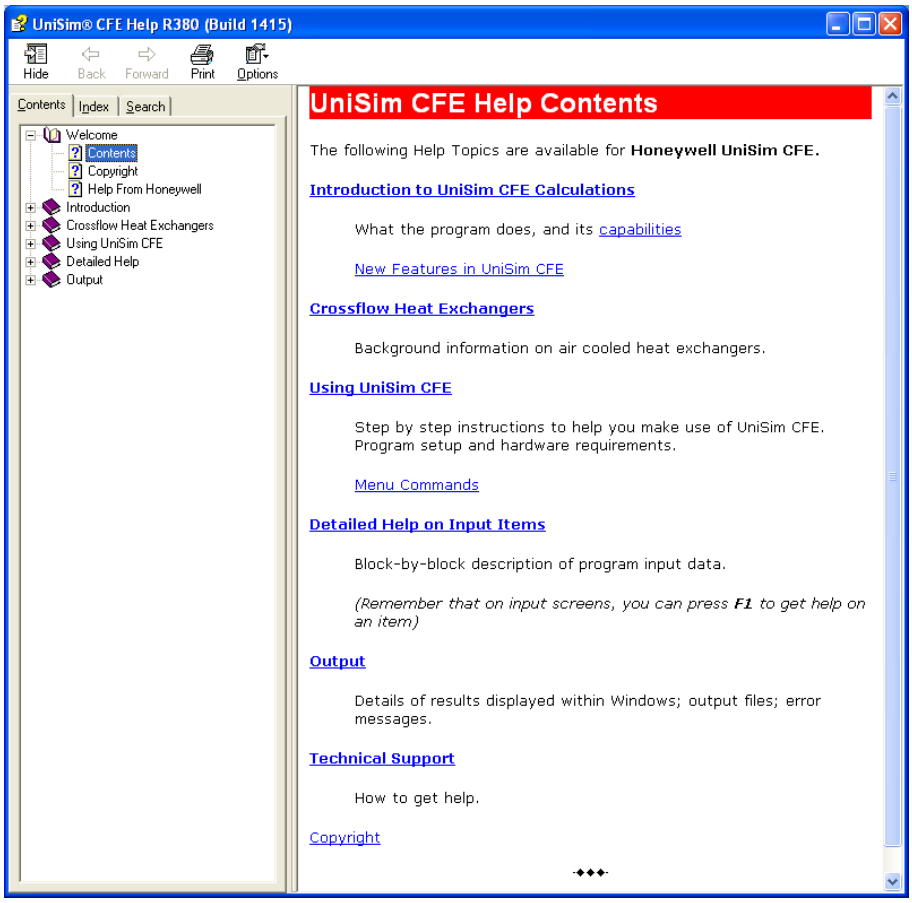
To load the Help when you are not running UniSim CFE, navigate to Program Files, Honeywell, UniSim Heat Exchangers Rxxx, UniSim CFE, **CFE.CHM**.

The Online Help is the most extensive documentation available for UniSim® CFE. It is available whenever you are running the program, or can be loaded separately. There are direct links to appropriate Help topics for every input item, and from many other places in the program.

The Help provides information on data input and defaults, navigating through the Interface, the Program Output, and on Errors and Warnings. There is also general information on crossflow heat

exchangers, and the reasons for choosing particular design features as shown below.

Figure 1



You will also find information on UniSim® CFE capabilities, new features in the latest version, and contact points for user support.

DR61 is the Design Report detailing the technical methods used in UniSim CFE.

The technical methods used in UniSim® CFE are proprietary. These methods are described in Design Report DR61.

Technical Support

Technical support is available by phone (1-403-509-1379 or 1-866-392-8748 toll free in North America), fax (1-403-216-2801).

E-mail support for customers with a current support contract for their product is available.

Honeywell	Email Address
Global	unisim.support@honeywell.com
North America	unisim.support@honeywell.com
Latin America	unisim.support.lar@honeywell.com
Europe, Middle East, Africa	unisim.support.emea@honeywell.com
Asia Pacific	unisim.support.ap@honeywell.com

On-line support can be accessed via http://www.honeywell.com/ps.

When contacting us via email or phone, please include in your message:

- Your full name, company, phone and fax numbers.
- The software version you are using (shown in the Help menu, About UniSim…).
- A detailed description of the problem (attach a simulation case if possible).

1 Getting Started

1.1 Introduction

Included with your **UniSim® Crossflow Exchanger Modeler** (*referred to as **UniSim® CFE***) software are a number of example input files that will familiarize you with the program. The cases are fully defined and ready to run. You can simply open the cases and run UniSim® CFE to see the type of output that can be calculated. This Getting Started Guide will step you through one of these example cases as a brief introduction to the UniSim® CFE architecture, input options, and available output information.

A complete set of results for the sample input files is provided in a separate location for quality assurance purposes in **Chapter 2 - Testing the Installation**.

A set of tutorials showing you how to set up input files for specific problems is included in **Chapter 7 - Examples**, of the Reference Guide.

1.2 Calculation Modes

Before examining and running the example input files, it is important to define the main types of calculations that UniSim® CFE can perform. These allow you to calculate the performance of air cooled/crossflow heat exchangers. The examples in the Reference Guide will elaborate on this, but for now the calculations will be described in the following sections.

1.2.1 Design

The program offers a range of exchanger widths and tube lengths, which satisfy the required thermal duty for a given X-side pressure drop and tube and fin geometry. The designer can select a design subject to constraints of maximum tubeside pressure drop and maximum and minimum tubeside velocities.

1.2.2 Checking

UniSim® CFE now offers a checking option which **c**alculates a heat transfer area ratio (actual/required) to indicate whether a given exchanger will meet a required duty, calculated from the tubeside flowrate and inlet and outlet conditions. An area ratio equal to or greater than unity indicates that the exchanger **will** meet the required

duty.

1.2.3 Simulation

UniSim® CFE offers nine simulation options:

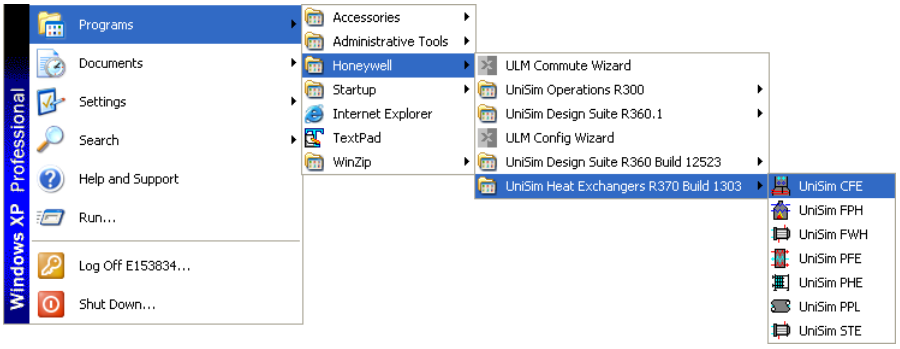
- **Standard simulation.** Calculates the tubeside outlet temperature from the tubeside inlet temperature and flowrate, and the X-side inlet temperature and flowrate. Similar to the tubeside outlet temperature option described below, but uses a combined forward and backward iteration technique.
- **Tubeside outlet temperature.** Calculated from the tubeside inlet temperature and flowrate, and the X-side inlet temperature and flowrate. Similar to the Standard Simulation described above, but uses a purely forward iteration technique.
- **Tubeside inlet temperature.** Calculated from the tubeside outlet temperature and flowrate, and the X-side inlet temperature and flowrate.
- **Natural convection.** Calculates the X-side flowrate during natural convection. Can be used when all fans in an air-cooled heat exchanger are switched off.
- **Tubeside flowrate.** Calculated from the tubeside stream inlet and outlet temperatures and the X-side inlet temperature and flowrate.
- **X-side flowrate.** Calculated from the specified tubeside conditions and the X-side inlet temperature.
- **Tubeside fouling resistance.** Calculates a hypothetical tubeside fouling resistance to give a process duty that matches the duty calculated from the input tubeside conditions. This gives an indication of the surplus heat transfer surface area available.
- **X-side face velocity.** Calculates the X-side mass flowrate from a given X-side face velocity (at actual conditions) and the bundle geometry and then carries out a Standard Simulation (see above).
- **Bundle pressure drop.** Calculates the X-side mass flowrate from a given bundle pressure drop (at actual conditions), inlet X-side stream density and the bundle geometry.

1.3 Example 1

In this first example we will take a brief look at how an existing dataset can be reviewed, run and the outputs accessed.

1. Start UniSim® CFE. This can be done several ways and will depend on how you set up your desktop. However, the two main ways are:
   - Click the **Start** menu, then **Programs**, **Honeywell**, **UniSim Heat Exchangers Rxxx** and **UniSim CFE** menu as shown below.

Figure 1.1

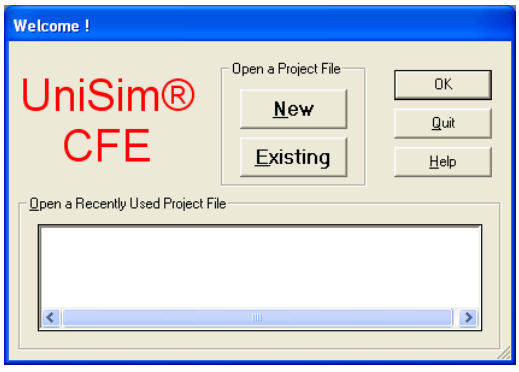


   - Use Windows Explorer and navigate to the CFE executable file.

After UniSim® CFE loads, you will see the main UniSim® CFE window and over this is the **Welcome** view. From this view you can either create a **New** file or open an **Existing** file. (If you have used UniSim® CFE previously, the project file you have worked with will appear in the **Recently Used Project File** list, making it easy to get back to files you were recently working on.)

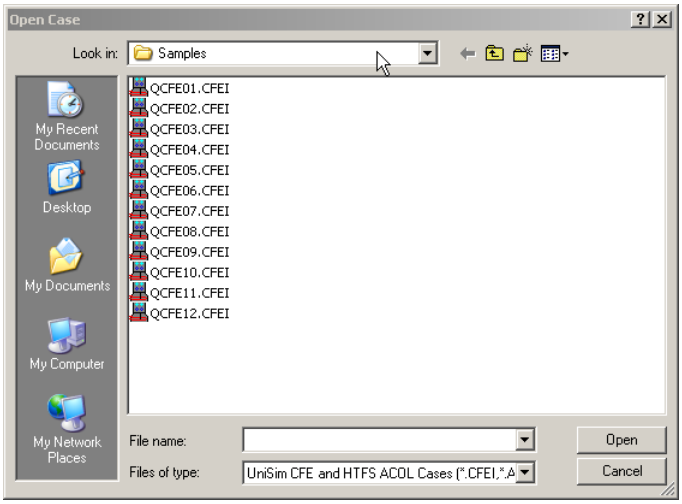
2. Click the **Existing** button, to select an Existing file as shown below.

Figure 1.2



3. The Open Case view appears as shown.

Figure 1.3



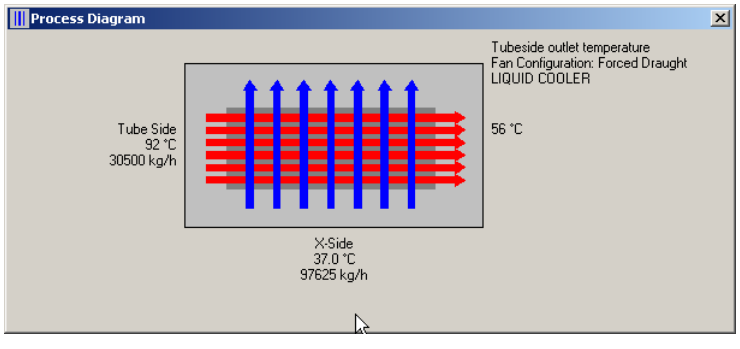
4. Navigate to the following directory:

   `C:\Program Files\Honeywell\UniSim Heat Exchangers Rxxx\UniSim CFE\Samples`.

(This is the default directory, the exact location may be different if you changed the UniSim® CFE destination directory during installation.)

5. Select the file **QCFE01.CFEI**. When the file has loaded, the Process Diagram view appears within the UniSim® CFE window as shown below.

Figure 1.4



If you cannot remember where a file is located, there is a Find File utility to help you. Select Find File from the File Menu or use the keyboard shortcut by pressing CTRL F.

To open other files, select **Open** from the **File** menu.

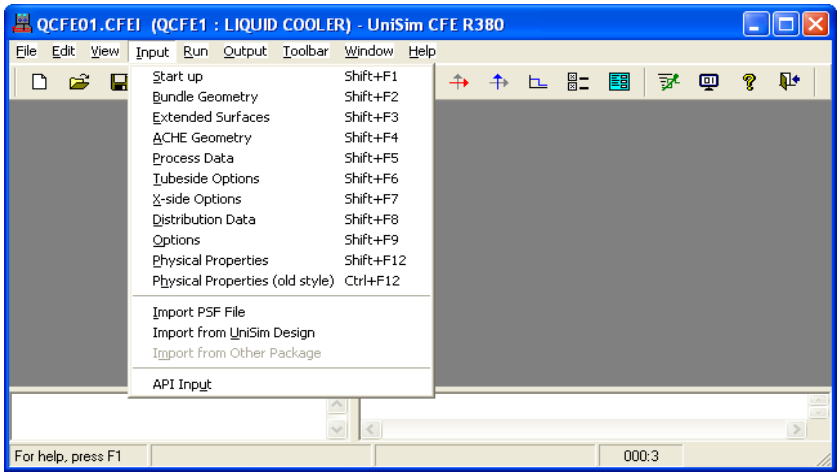
Open icon

For most common activities there are shortcuts. To open a file, you can either click on the Open icon, or use the keyboard shortcut by pressing CTRL O.

To use the Welcome view again, select **Start Project** from the **View** menu. (Note however, that you can only have one project active at a time.)

1. Click on **Input** in the menu bar.

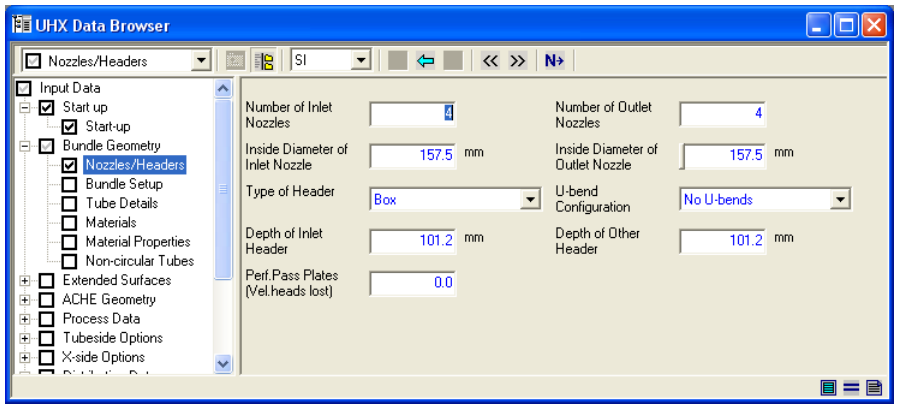
Figure 1.5



The Input menu gives access to all of the input data. The menu itself is divided into the different types of data you need to describe the heat exchanger and the operating conditions. These include different aspects of geometry, process conditions and physical properties.

2. Select Bundle Geometry from the Input menu. The Bundle Geometry view appears as shown below.

Figure 1.6



Bundle Geometry icon

You should see values which give the basic bundle details such as nozzle and header data. This view is typical in that the data is entered in either a text box or via a drop-down list. The drop-down list shows

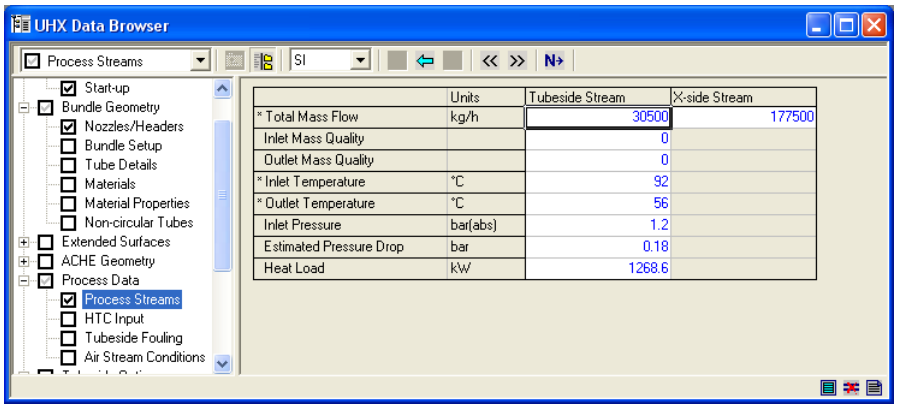
the list of possible values that you can enter.

If at any point you are not sure what input you want or something is not clear, you can press F1 and get context sensitive help. (i.e., if you select the Type of Header and press F1, you can see a list of the Header Types.)

3. Look at the process data by selecting **Process Data** from the **Input** menu, by clicking on the **Process Data** button or by clicking on the **navigation tree**. The Process Data view appears as shown in the following figure.

Process Data icon

Figure 1.7



4. This shows another view where the input items are arranged in the form of a spreadsheet. If the data do not fit on the view, a scroll bar allows you to access the other input items. The spreadsheet view is used when data are required several times (in this case, for the two streams in the exchanger).

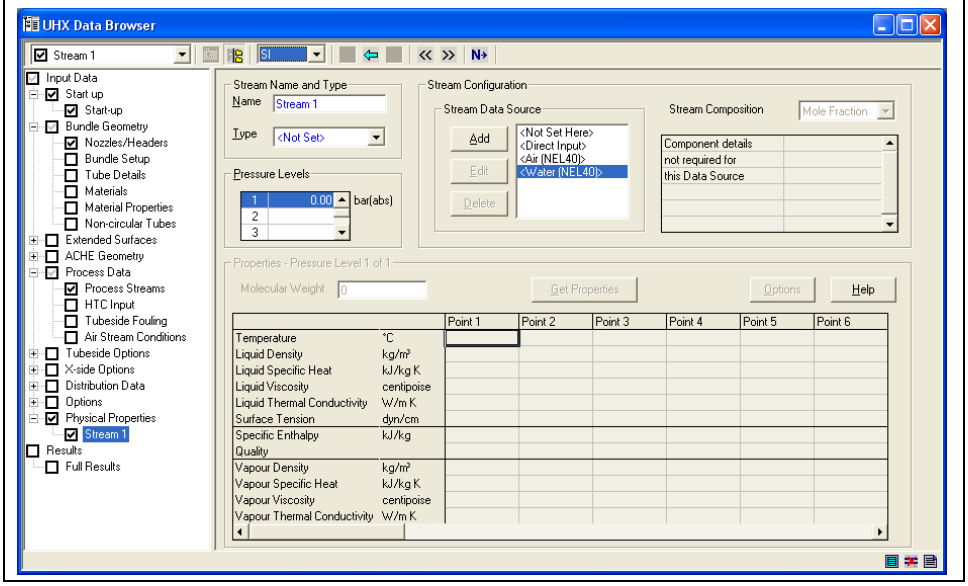
The left column is for the Tubeside Stream, and the right column is for the X-side Stream.

5. Finally, look briefly at the physical properties input by selecting **Physical Properties**.

The top level information about each stream is shown in the following

figure.

Figure 1.8



Depending on the type of physical property data you are working with, you can either enter the property data for the stream directly or enter data for components and allow UniSim® CFE to perform vapour liquid equilibrium and mixture calculations. All of the physical property data are managed through these views.

Since this is an existing case, all the necessary data has already been entered.

6. Run UniSim® CFE by doing **one** of the following:

Run icon

- Click on the **Run** icon in the tool bar;
- Open the **Run** menu and select **Calculate All**;
- Press **F4**.

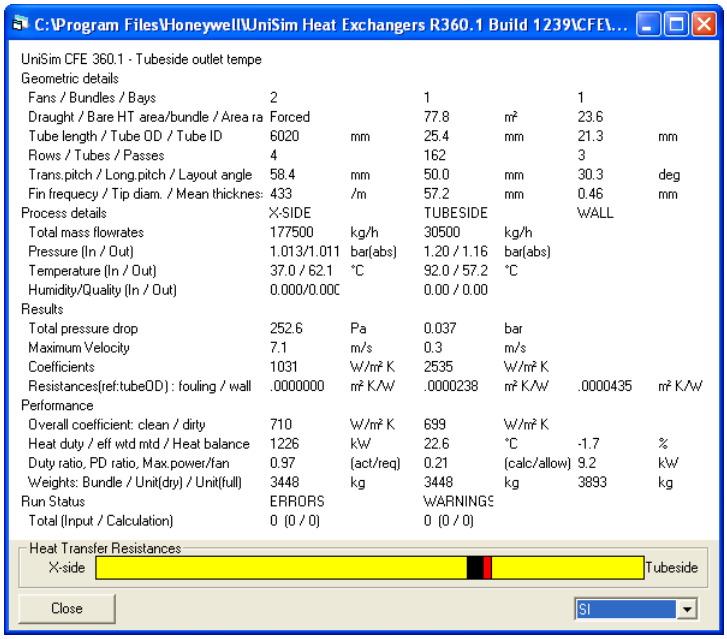
UniSim® CFE now displays a status window that reports the progress of the run.

When the run is complete, there are three possible outcomes and corresponding outputs appear.

- Successful run with no fatal errors and no warnings. A view showing the Results Summary appears.
- Successful run with no fatal errors but with one or more warnings. The Results Summary appears with the Error/ Message Log which contains a description of the warnings that occurred.
- Failed run due to fatal errors. The Error/Message Log is shown with a description of the errors that occurred.

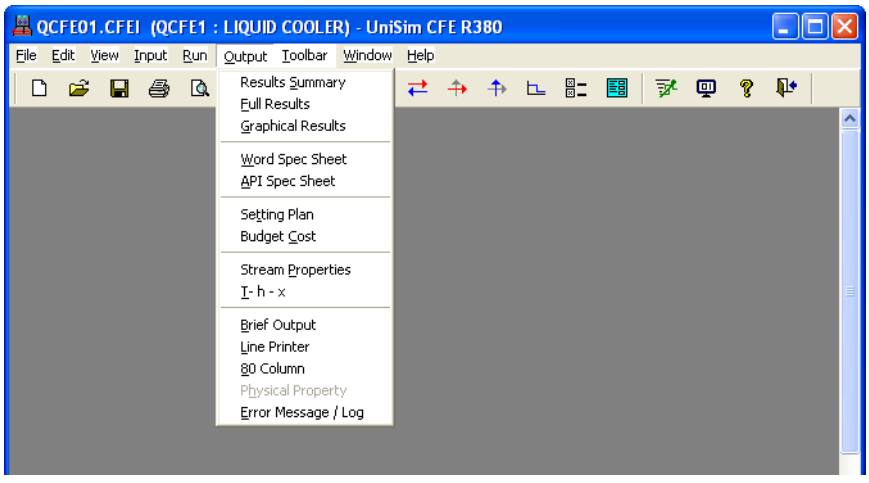
For this case, you will see the Results Summary which shows important results as shown below.

Figure 1.9



There are many different outputs that can be viewed from the **Output** menu as shown below.

Figure 1.10



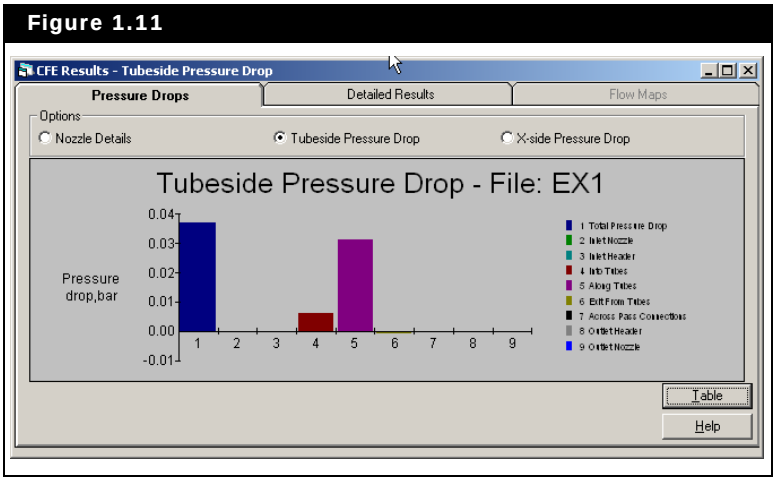
Now you will look at the different types of output available.

The first group of outputs are all special “Windows” outputs (views of the data that are in the form of graphs or tables):

1. Open the **Output** menu.
2. Select **Graphical Results**.
3. Select either **Pressure Drops** or **Detailed Results** views of the exchanger components.
4. Select **Tubeside Pressure Drop**.
5. Select **Chart**.

This produces a graph of pressure drop through the tubeside components as shown below. This will alert you to any unusual behaviour such as a high nozzle pressure drop.

Figure 1.11

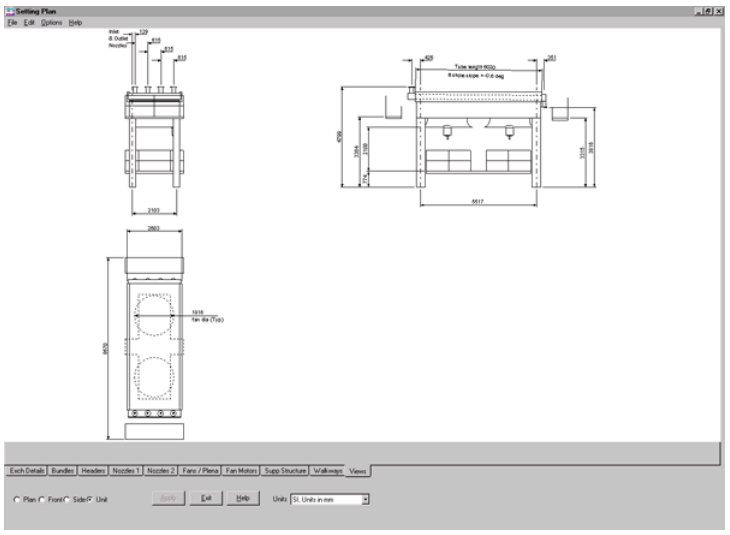


Next there are outputs in the form of an API Specification Sheet. The first of these requires that you have Word for Windows available (this must be setup via **Preferences** from the **File** menu). The second is a view of the built in API sheet. (You will look at this further in the second example.)

If you want to know how your exchanger may appear, the Setting Plan

can give you various views such as that shown below.

Figure 1.12



You can also get a budget estimate of a capital cost of a complete air-cooled heat exchanger unit by selecting **Budget Cost** from the output menu.

The next group of outputs offers the facility to view the physical properties of each stream in tabulated format and perform various plots. It also offers tabulated information on temperature-enthalpy-quality (THX) data and stream composition.

The final group of files are outputs generated directly by the calculation engine. They are all text files and contain different aspects of results generated. Of these, the one that you are most likely to use is the Lineprinter Output which contains all of the important information for your run.

1.4 Example 2

In the first example, you saw that there is a lot of data that can be entered for UniSim® CFE which demonstrates its full power. It is possible to reduce the amount of input you must consider in two ways:

- Basic mode
- Interactive API Input Sheet

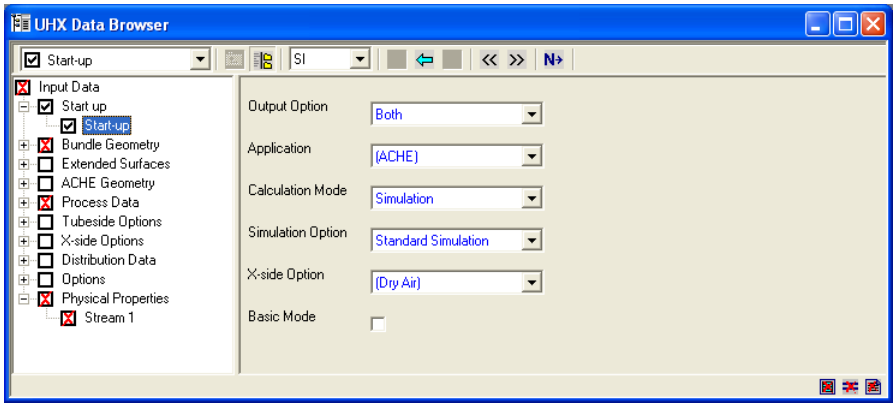
To demonstrate data input through Basic Mode:

1. Open the **Window** menu and select **Close All** to close all the open

views.

2. Open the **File** menu and select **New**. The Start up view appears as shown below.

Figure 1.13



All the main input views have keyboard shortcuts; for instance, the Start up view is SHIFT F1; and Physical Properties Data is SHIFT F12.

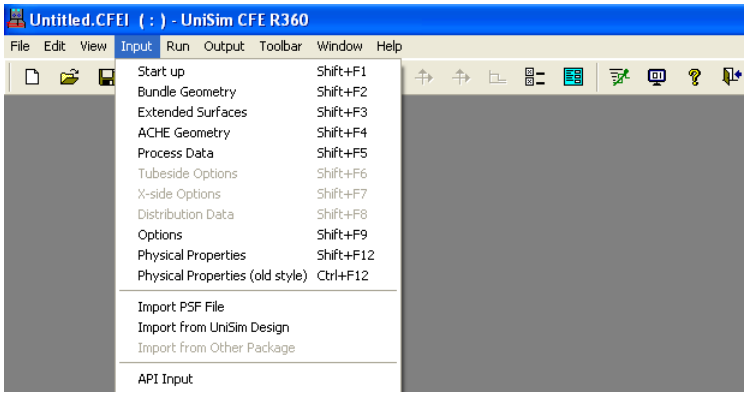
The Start up view (shown in **Figure 1.13**), also appears when you create a new file. It is here that you must make a decision about what type of calculation will be performed. You will see that Calculation Mode is currently set to Simulation; the Simulation Option is set to Standard Simulation, and the X-side Option is set to Dry Air.

3. Check the **Basic Mode** checkbox and click **OK**.

In this input mode you will see the information needed to model the heat exchanger, but many of the special features, options and capabilities are hidden. To demonstrate this, click on the **Input** menu (as shown below) and you will see some of the input items are now

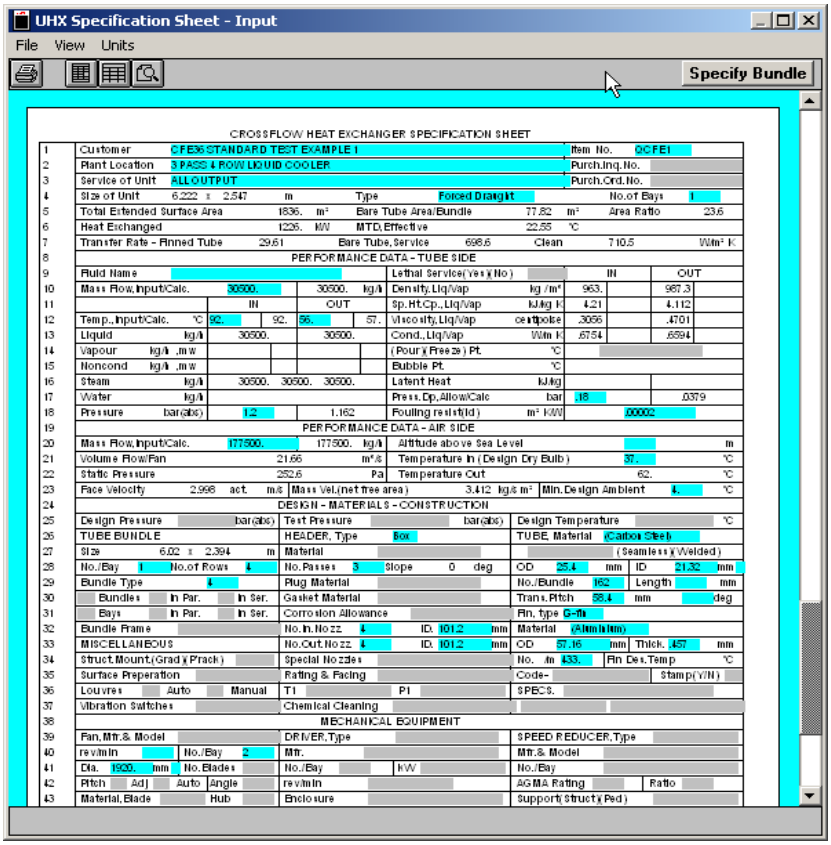
hidden in comparison with **Figure 1.5**.

Figure 1.14



4. Now select **API Input** from the **Input** menu. The API Input view appears as shown below.

Figure 1.15



Many of the fields are blue. These are input fields and are directly connected to the main input forms. You can make changes here and they will be reflected in the main inputs. If you are creating a new case and you have data available in the form of an API Sheet, this method may speed up your data input.

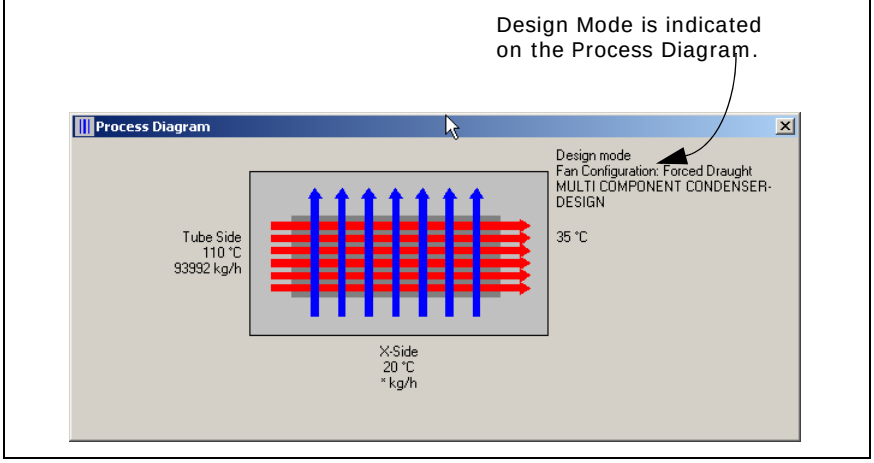
Additional fields in grey allow the addition of comments or items that are not actually required for UniSim® CFE to run but may be useful in completing an API Sheet that can be printed.

1.5 Example 3

The main objective of this example is to demonstrate the UniSim® CFE **Design Option** capability.

1. Use **Close All** from the **Window** menu to close all the current views.
2. Select **Open** from the **File** menu to open the design example file **QCFE07.CFEI** from the default directory **\Program Files\Honeywell\UniSim Heat Exchangers Rxxx\UniSim CFE\Samples**. The Process Diagram view appears as shown below.

Figure 1.16



By choosing the design option, the number of input items required for your exchanger are reduced (i.e., the input for **Bundle Geometry**). For example, in this case the **Nozzles and Header** details will disappear and other items will need less input data.

The most important input data required for the **Design Option** are **Design Parameters** for **ACHE Geometry** and **Process Data** where minimum and maximum values for these **Design Parameters** are defined to produce a range of exchanger designs.

Run icon

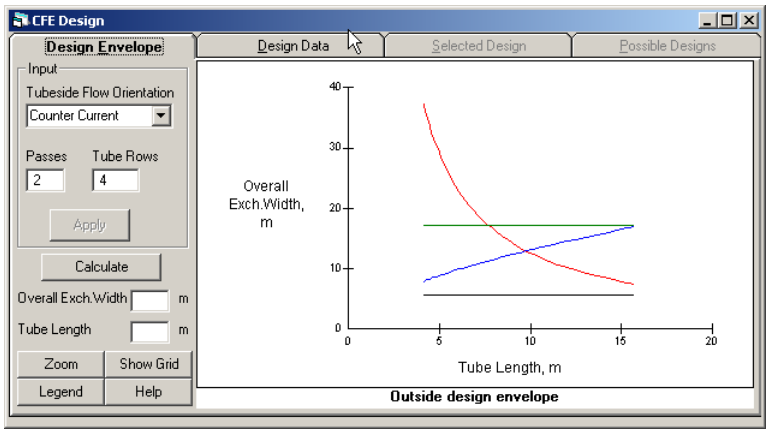
3. Open the **Run** menu. Select **Calculate All**.
4. Click on the **Run** button, or press **F4** to run the design. The UniSim® CFE Design Envelope appears as shown in the following figure.

Figure 1.17

The details of the design are shown under the Selected Design tab and are calculated from the information given under the Design Data tab. When a design has been selected, it can be refined by running a simulation.

The Possible Designs tab will show you other designs for the total exchanger width and tube length selected. The selected design has the lowest total number of fans and bundles.

When the mouse cursor is within the design envelope, the overall exchanger width and tube length are shown for the current location at the bottom left of the screen.



Select a design by clicking in the Feasible Design Area. When the cursor is within this area, the text “Feasible Design Area” appears in the status bar at the bottom of the graph, and the cursor shape changes to a cross-hair.

The Design Envelope consists of four lines:

- **Thermal duty line** which shows all of the heat exchanger sizes (overall exchanger width and tube length) that will satisfy the thermal duty;
- **Maximum tubeside pressure drop line** which shows the locus of all points where the tubeside pressure drop is equal to the maximum specified;
- **Maximum & minimum tubeside velocity lines** which show the relationship between the overall exchanger width and the tube length for given maximum and minimum tubeside velocities.

2 Testing the Installation

2.1 Introduction

A set of twelve sample UniSim® CFE cases, including both input and output files are provided with UniSim® CFE for Quality Assurance (QA) purposes. As a check that you have installed UniSim® CFE correctly, you should run the input files and compare your results files with those provided.

UniSim® CFE files have a file extension **.CFEx** (where x is an indicator of the type of file (input or one of the various outputs). A full listing is given in the Online Help.

When you installed the software, the QA files are stored in a subdirectory (QADATA) of the directory containing the main UniSim® CFE folder. The twelve sample cases have file names **QCFE01** to **QCFE12**, and file extensions are **.QAx** instead of **.CFEx**. The different extensions are used to ensure that you do not accidentally overwrite the QA files when running UniSim® CFE.

UniSim Heat Exchanger programs are now installed to **\Program Files\Honeywell\UniSim Heat Exchangers Rxxx**.

Copies of the twelve QA input files, with the standard input file extension **.CFEI** are installed in the ....**\UniSim Heat Exchangers Rxxx\UniSim CFE\Samples** directory by the installation procedure.

2.2 Creating Output

The first test on the installation is with QCFE1:

1. Copy the **QCFE01.QAI** file from the ....**\UniSim Heat Exchangers Rxxx\UniSim CFE\QADATA** directory to another directory (e.g., ...\My Documents\My UniSim Heat Exchanger Cases).
2. Rename the file, and give it the extension **.CFEI** (for example MYTEST1.CFEI.
3. Start UniSim® CFE. On the view that appears, click on Existing. Navigate to

   `My Documents\My UniSim Heat Exchanger Cases\MYTEST1.CFEI.`
4. UniSim® CFE loads the case.
5. Run UniSim® CFE with this case.
6. Compare the results files from your run with the results files supplied with UniSim® CFE.
7. Checks may be repeated with the other QA files supplied.

2.3 Comparing Outputs

Your calculated results are files named **MYTEST1.CFEx**, in directory **\My Documents\My UniSim Heat Exchanger Cases** and these need to be compared with the supplied results files **QCFE01.CFEx** in directory ....**\UniSim Heat Exchangers Rxxx\UniSim CFE\QADATA**.

Remember, the **QADATA** files supplied with UniSim® CFE have the extension **.QAx**.

The most important comparisons are the **.CFEV** and **.CFEL** files but other files can be compared as well. The **.CFEV** file is the **Results Summary** and is a relatively short file. You can do the comparison using a file difference utility, or by printing off the two files and looking for differences.

If the files are identical, (except for the run time and input file name recorded in the output), the QA check is successful. If the files differ slightly, but only in the fourth or fifth significant figure of one or two variables, the QA check on this example is also successful. If there are more significant differences, consult Honeywell.