

# Introduction to HYSYS Simulation

## Heat Transfer Calculations

Chemical Engineering, Rowan University (Revised 4/18/2001)

In this exercise, you will simulate two heat exchangers and calculate the heat transferred between a hot and a cold stream using a chemical process simulation package from Hyprotech Inc. This program is used by industry to design and simulate process plants such as oil and gas refineries, chemical and pharmaceuticals production facilities. A typical sampling of well-known companies using this software are given below:

- DuPont, Eastman Kodak, Monsanto, Hoechst Celanese, Rohm & Haas
- Shell, Amoco, Exxon Chemical, Olin, Conoco, Phillips
- Foster Wheeler, Arthur D. Little, Ballard, Koch
- Hoffman LaRoche, International Flavors and Fragrances,

This software is written in C++ which is a language that you have learned. The cost of this software is approximately \$45,000 for the dynamic version and \$20,000 for the steady-state version. This price is for only one user! For more information on Hyprotech go to [www.hyprotech.com](http://www.hyprotech.com)

The overall process for this simulation is to

- Select a thermodynamics package that describes the physical and chemical properties of the chosen chemicals
- Select the unit operations
- Define all required inputs and output streams. Specify required values of temperature, pressure, flowrate and chemical concentration.

### Start your computer:

Rowan University has a license to run HYSYS PLANT version 2.2, which is both a steady state and dynamic simulation program.

For convenience Hyprotech suggests that you copy the HYSYS directories to your personal drive on galaxy. These are located in [\galaxy\public\chemical\engineering\HYSYS](http://\galaxy\public\chemical\engineering\HYSYS) (To do this you may need to map a drive (Start, Windows Explorer, Tools, Map Network Drive...)) In these directories you will save your configuration files and cases.

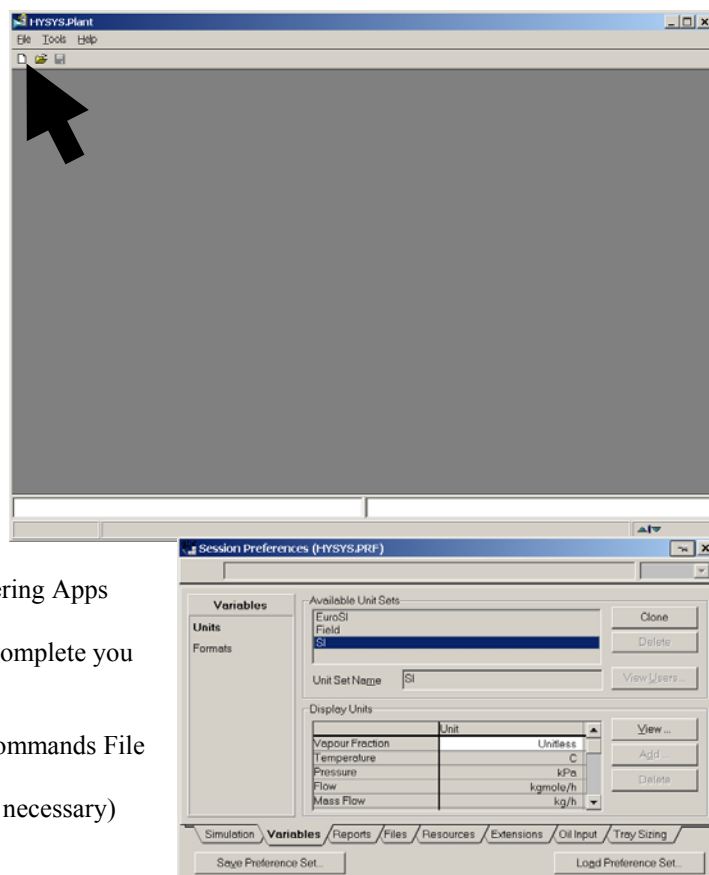
In HYSYS there is a limited online **Help** package, which can be obtained using the Help command from the menu bar or by pressing the F1 key. There is an extensive collection of manuals that can be accessed from [\engnet1\apps\Hysys\\_Doc\Menu.pdf](http://\engnet1\apps\Hysys_Doc\Menu.pdf). Remember that we are running HYSYS PLANT.

1. Start HYSYS PLANT  
**Third Floor Lab** (windows NT) Start => Engineering  
Network apps => Engineering Apps => HYSYS Plant  
**First floor labs** (windows 2000) Press Start => Engineering Apps  
=>HYSYS Plant

This package will take about a minute to start up and when complete you will see the screen given above.

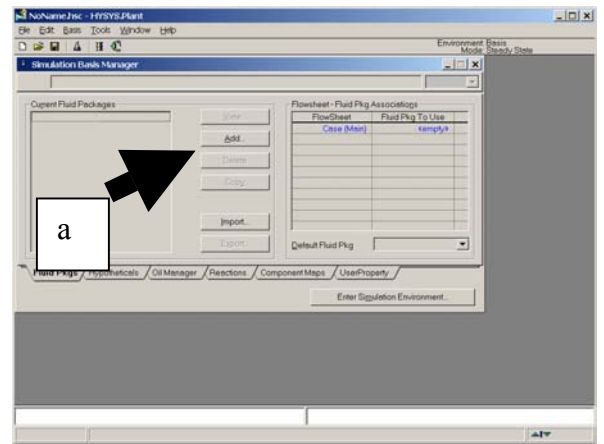
To start a new case click on the white page icon or use the commands File New Case. See previous page.

2. Change your units from American engineering to SI: (if necessary)
  - 2.1. Go to the command **T**ools, **p**references
  - 2.2. Choose the Units tab



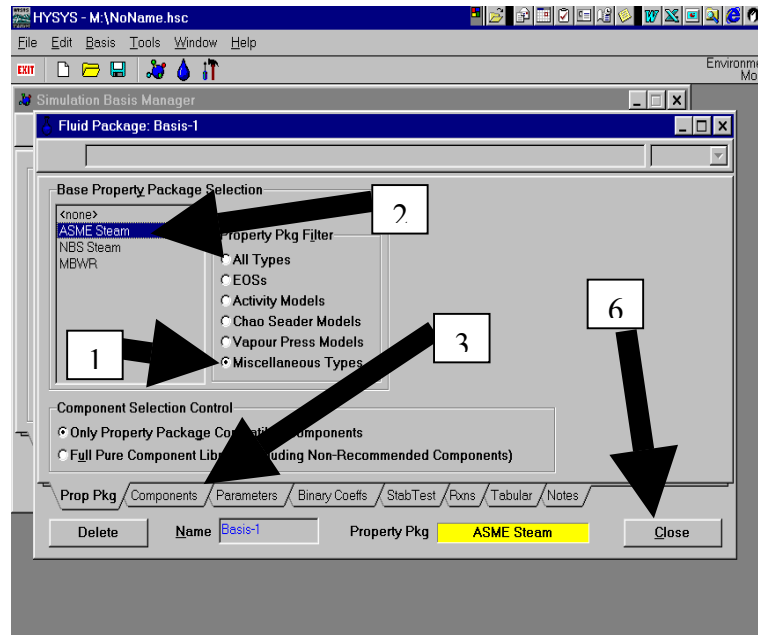
- 2.3. Select SI
- 2.4. Press Save Preference Set to your galaxy drive
- 2.5. Press close.

The result will be the following menu screen to choose a thermodynamic property package. You will learn more about these packages in your chemical engineering thermodynamics course. We will use the ASME Steam as our property package.



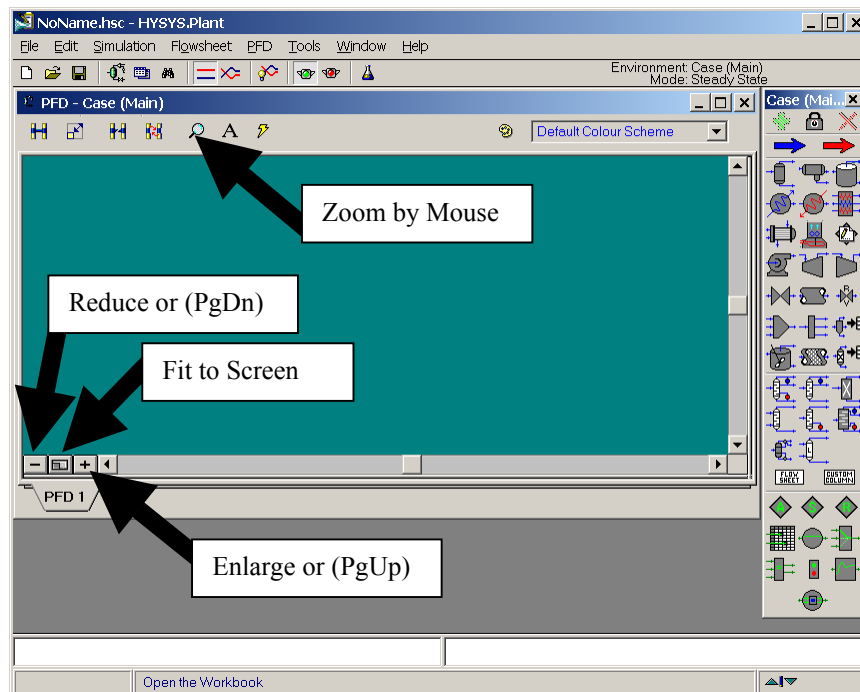
3. Choose the Property package and chemical components

- 3.1. Choose Miscellaneous Types
- 3.2. Choose ASME
- 3.3. Press the Components tab
- 3.4. Add *water* by selecting the component and pressing the Add Pure button
- 3.5. Press the x or close button for *this screen*
- 3.6. Press the Enter Simulation Environment... Button (see figure above)

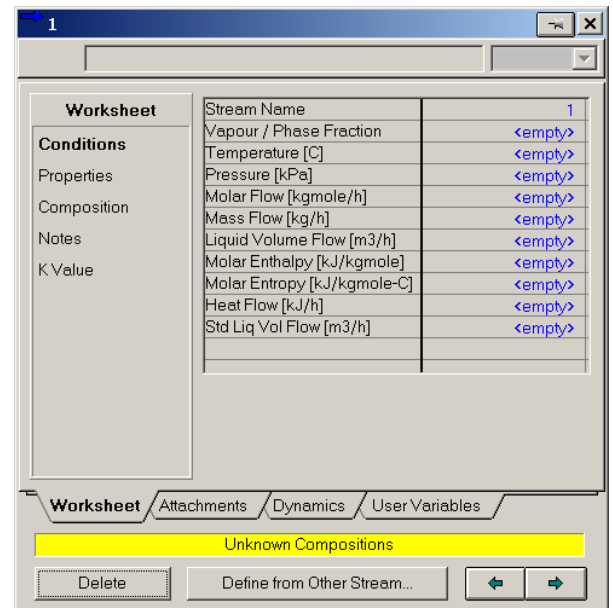
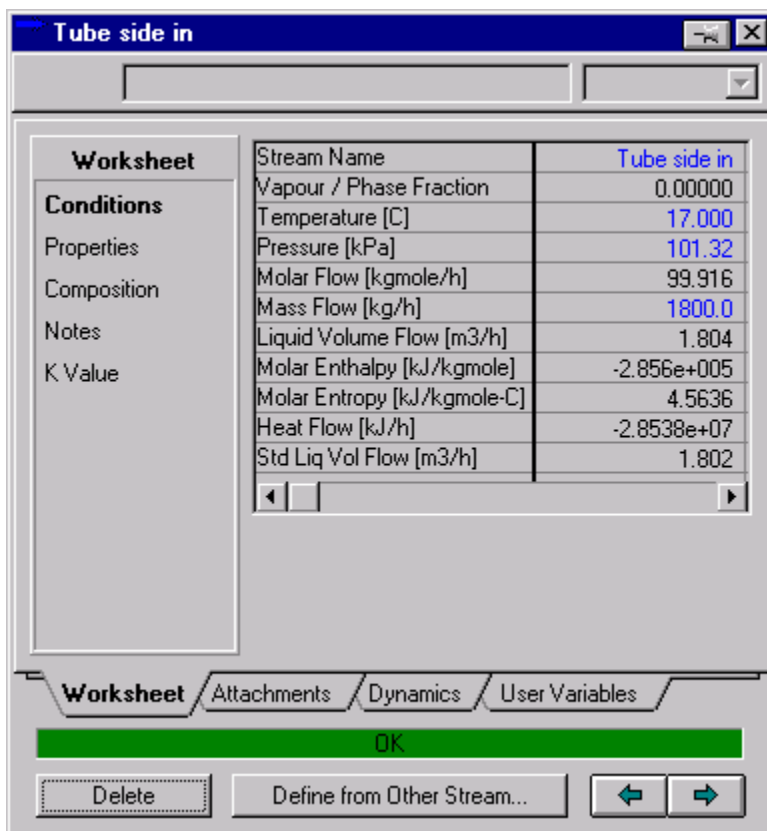


You should now see the following screen titled PFD - Case (Main). In the green screen you will construct a process flow diagram (PFD) of a heat exchanger. The pictures of equipment shown to the right of the green screen contains the equipment that you will add.

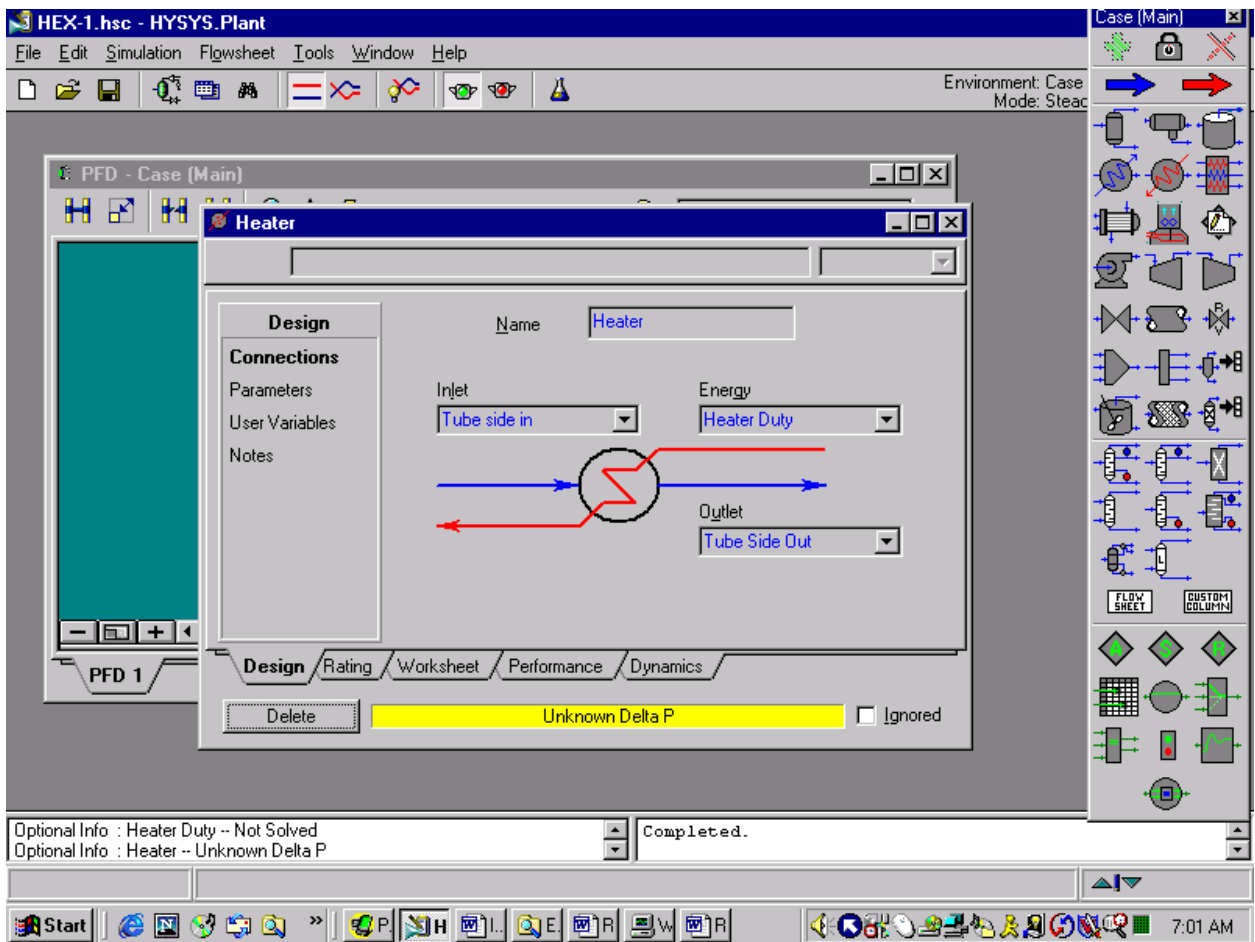
4. Save the file on your galaxy drive. File, Save as, and change the path to your personal galaxy drive. I would suggest always saving your files with a unique descriptor (your name). The files are always printed with your filename showing on the page. To determine which letter designates your galaxy drive, open Windows Explorer and find the letter corresponding to your “galaxy” drive.
5. Select a material stream by double clicking on the blue arrow and the menu labeled 1 will show on the pfd screen. Notice that a yellow warning sign appears telling you what is required: “Unknown Compositions.” The other warning colors are red: “requires a feed stream,” and green for “OK” for it has been solved.



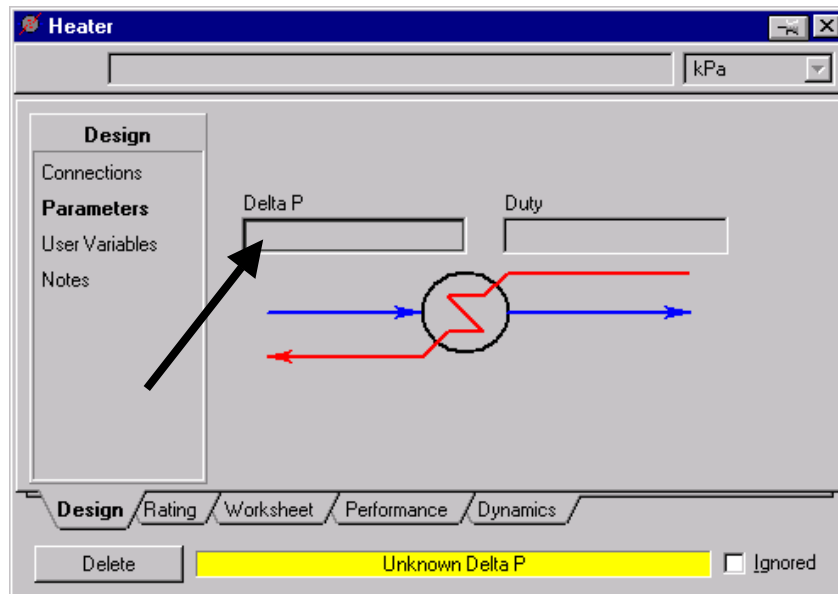
6. Fill out this menu as follows:
  - 6.1. Stream Name: Tube Side In
7. To enter your readings of temperature, pressure and flowrate for the stream just type in the values measured in the lab.
  - 7.1. Type in your value of pressure as 1 atm. Type in the number, a space and the letters atm. Hit return and notice that it converts the units to SI automatically!
  - 7.2. To specify mole fractions of the component do either:
    - 7.2.1. select the composition options on the left side and click on the Edit button or
    - 7.2.2. double click on the cell next to Molar Flow [kgmol/h]:
  - 7.3. Then type in 1.0 as the mole fractions of water.
  - 7.4. Check to see that you have 1.0 in the Total box, and then press the OK button.
  - 7.5. Type in the flow you measure in the lab.



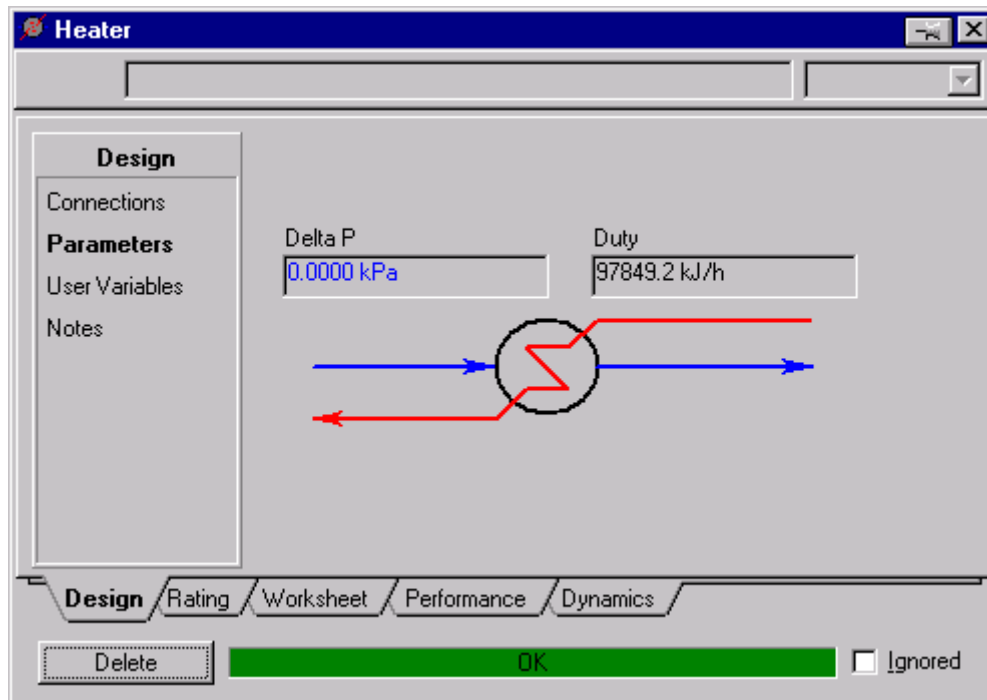
8. Next install a heater from the unit operations palette. To select and add the heater double click on the heater icon. The heater has the red-downward pointing arrow. Fill out this menu as follows for the heater:
  - 8.1. Name: Heater
  - 8.2. Inlet: Tube side In.
  - 8.3. Outlet: Tube Side Out
  - 8.4. Energy: Heater Duty.



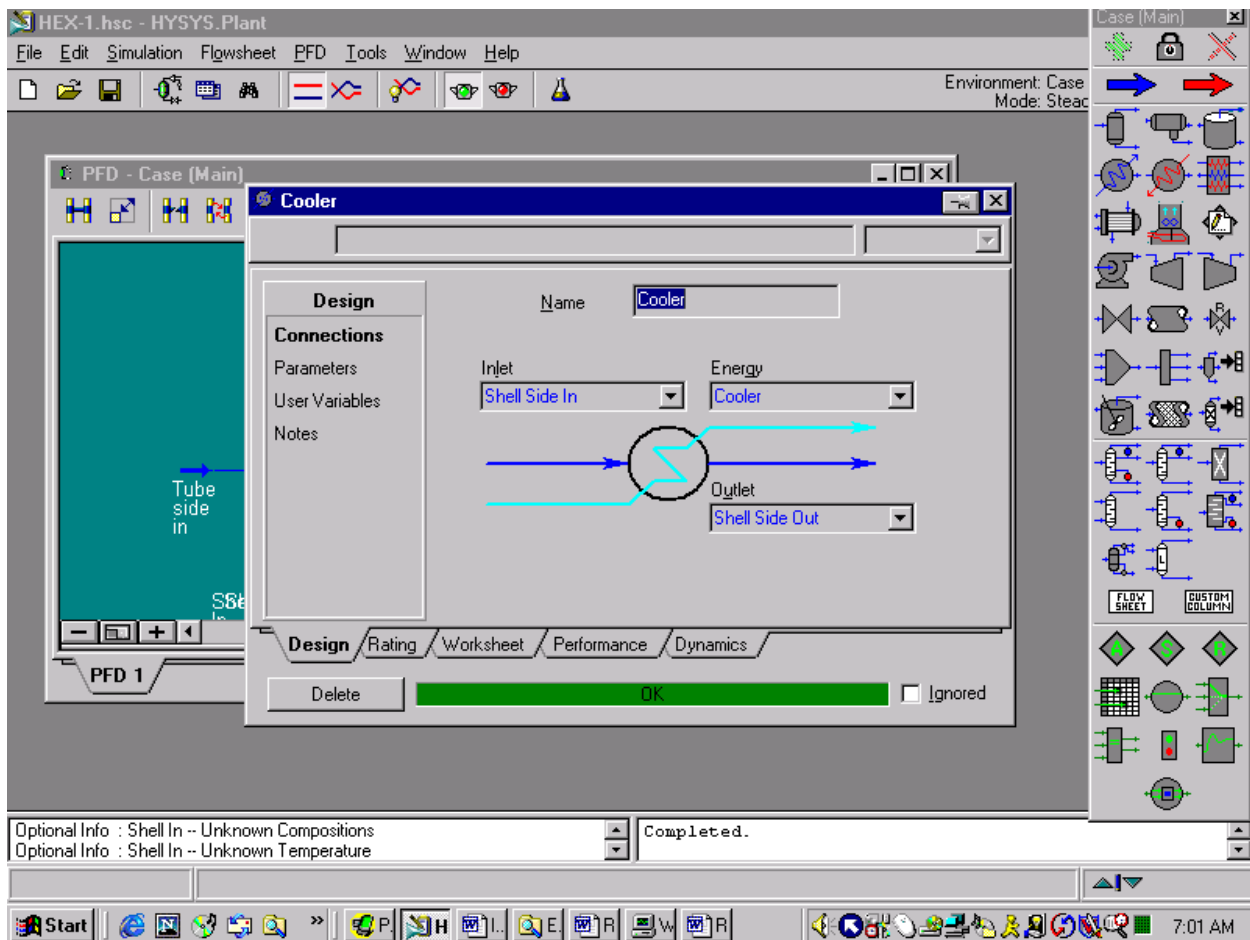
- Go to the Worksheet (click on the tab) and fill in the temperature of the Tube Side Out Stream=30 C



10. The box in yellow, the pressure drop through the heater is still unknown. Since you have no measurements for the pressure drop, we will enter zero (You will learn how to estimate this value in your Senior Year). Click on "Parameters" and enter 0.0 KPa.
11. Now your heater is fully defined and the Heat Duty has been calculated. HEAT DUTY \_\_\_\_\_



12. Add a Cooler. This cooler will represent the Shell Side of the heat exchanger.
  - 12.1. Double click on the cooler icon.
  - 12.2. Name: Cooler
  - 12.3. Inlet: Shell Side In
  - 12.4. Outlet: Shell Side Out
  - 12.5. Energy: Cooler Duty
13. Go to the Worksheet to specify the new material streams.
  - 13.1. Enter the flowrate, temperature, pressure and composition for the Stream: Shell Side In
  - 13.2. Enter the Shell Side Out measured temperature.
14. Go back to the Design layer by clicking on the Design tab and then click on Parameters to enter the pressure drop for the shell side. Once again, enter the value zero.



15. Copy the heat duty of the Cooler \_\_\_\_\_
16. The hot water in the Shell Side is responsible for heating up the cold water that runs in the Tube Side. Write an energy balance for the system.  $Q_{cold} = F_{cold} c_{p,cold} \Delta T_{cold}$ ,  $Q_{hot} = F_{hot} c_{p,hot} \Delta T_{hot}$ ,  $Q_{cold} = ? = Q_{hot}$

Compare the heat duties of both heat exchangers. Explain.

17. Printout your HYSYS workbook Specsheets and PFD. To print a specsheet:
- Check the printer by selecting File Printer Setup. The graphics printer prints the pfd's and the report printer prints text or workbook.
  - Select the workbook by pressing the toolbar button as shown in the figure below.
  - Select File, Print,
  - Choose All Pages - Compact.
  - Preview it

**Also printout your process flow diagram, (PFD) by choosing the print command while the PFD is active.**