KM 331 PROSES BENZETİM PROGRAMLARI
DERS NOTLARI [1-4]

Kaynaklar

1. Chemcad User Guide and Tutorial, Chemstations, Inc. Version 6.1.
2. Aspen Technology, Inc., Apsen HYSYS ® Version 7.
3. ChemCad Eğitim Notları , Chemstations, Inc- Houston,TX,USA.
4. A Guide for Getting Started in Aspen HYSYS
   Dinu Ajikutira, Sr. Director, Engineering Product Marketing, Aspen Technology, Inc.

Aspen HYSYS Proses Akış Diyagramının Çizimi

Once you have specified the components and fluid package, and entered the simulation environment, you will see the view as shown in Figure 1-7. Before proceeding, you should taking care of a few features of this simulation window:

1. HYSYS, unlike the majority of other simulation packages, solves the flowsheet after each addition/change to the flowsheet. This feature can be disabled by clicking the **Solver Holding** button (the **red** light button ) located in the toolbar (see Figure 1-7). If this button is selected, then HYSYS will **not** solve the simulation and it will **not** provide any results. In order to allow HYSYS to return results, the **Solver Active** button (the **green** light button ) must be selected.
2. Unlike most other process simulators, HYSYS is capable of solving for information both downstream and upstream. Therefore, it is very important to pay close attention to your flowsheet specification to ensure that you are not providing HYSYS with conflicting information. Otherwise, you will get an error and the simulation will **not** solve.

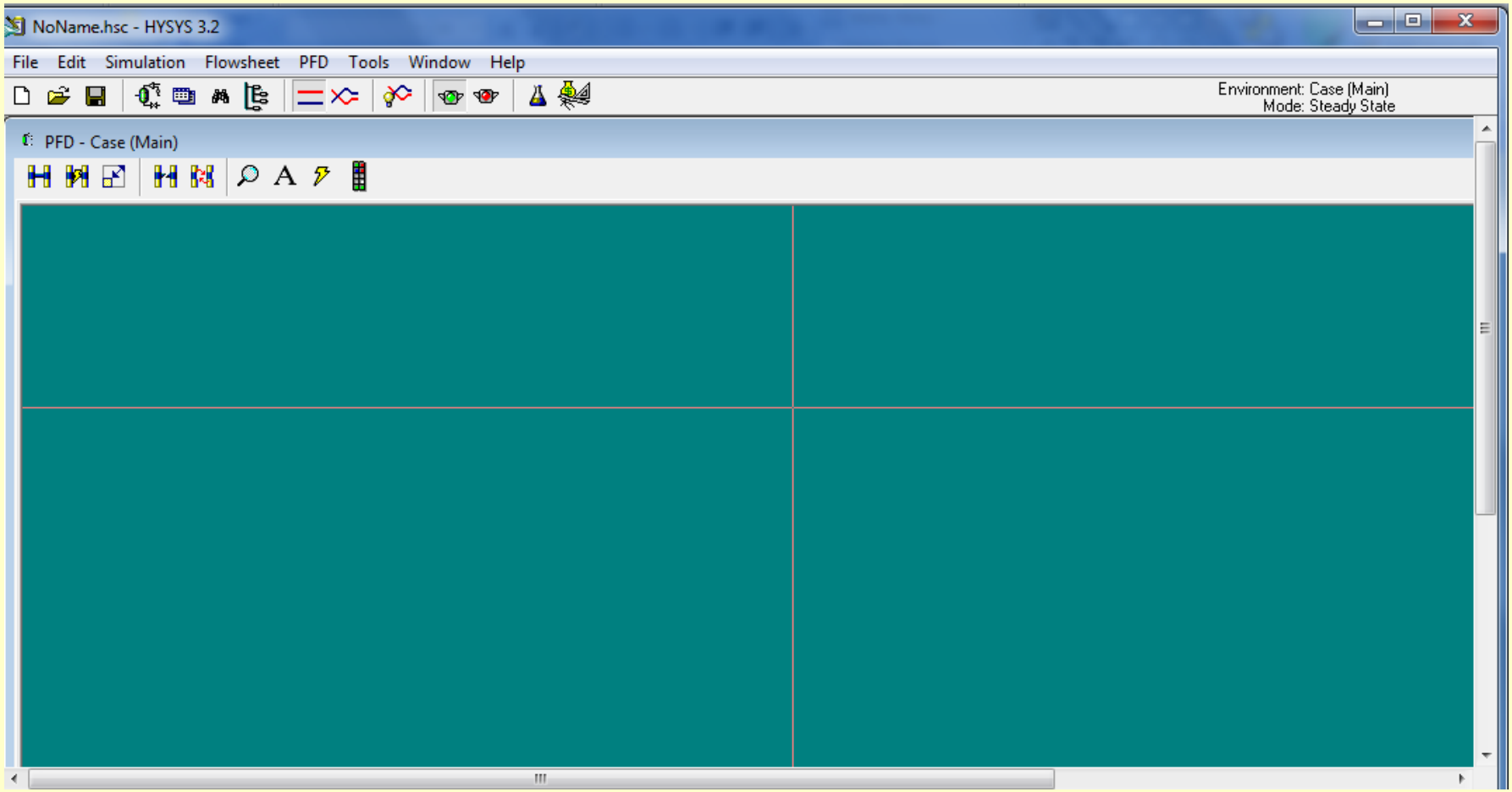
NoName.hsc - HYSYS 3.2
File Edit Simulation Flowsheet PFD Tools Window Help
Environment: Case (Main)
Mode: Steady State
PFD - Case (Main)

Figure 1-7

Re-Entering the Simulation Basis Manager

When the basis of the simulation has to be changed, the Simulation Basis Manager needs to be re-entered. Simply click on the  icon on the top toolbar to re-enter it.

Accidentally Closing the PFD

Sometimes, people accidentally click the **red X** on the PFD. To get it back, simply go to **Tools –> PFDs**, make sure **Case** is selected, then click **View**.

Object Palette

On the right hand side of Figure 1-8, you will notice a vertical toolbar. This is known as the **Object Palette**. If for any reason this palette is **not** visible, got to the **Flowsheet** pulldown

menu and select **Palette** or press **F4** to display the palette. It is from this toolbar that you will add streams and unit operations to your simulation.

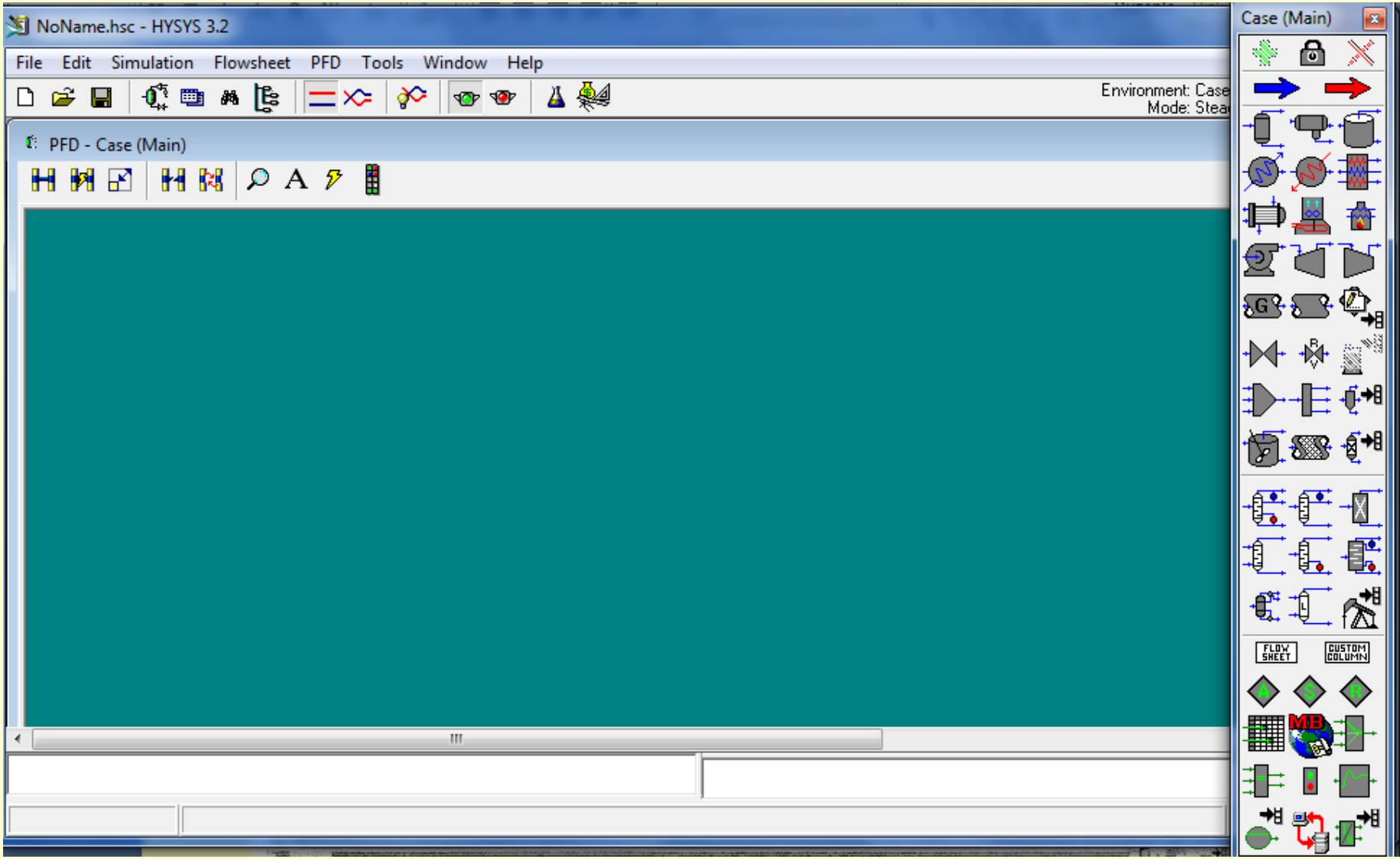

Figure 1-8

1.8 Adding Material Streams

Material Streams are used to transport the material components from process units in the simulation. A material stream can be added to the flowsheet in one of three ways:

1. Click on the blue arrow button on the Object Palette
2. Selecting the "Flowsheet" menu and selecting "Add Stream"
3. Pressing F11

Using any of the above methods will create a new material stream (a Blue arrow) on the flowsheet, refer Figure 1-9. The HYSYS default names the stream in increasing numerical order (i.e. the first stream created will be given the name "1"). This name can be modified at any time.

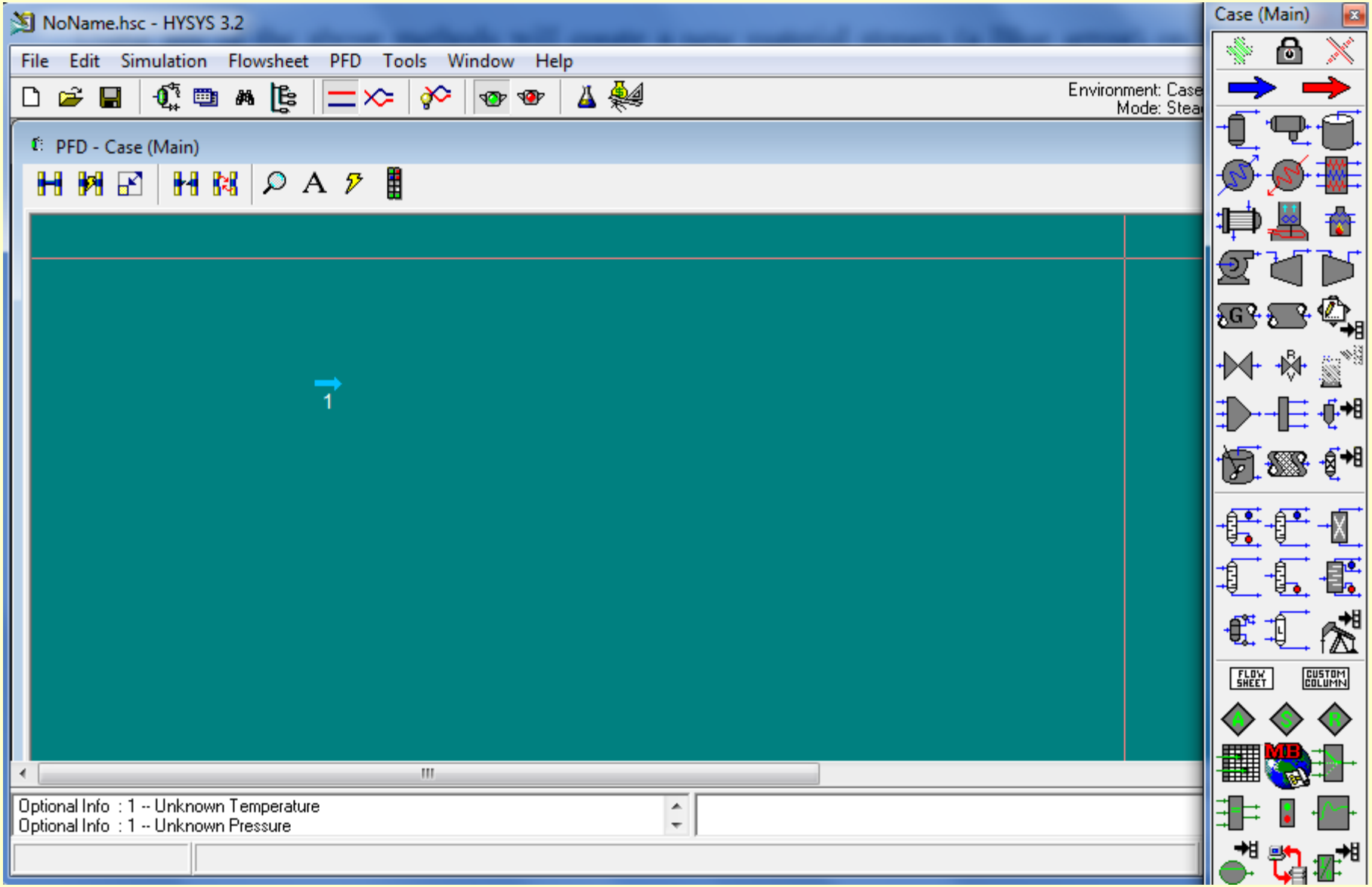
NoName.hsc - HYSYS 3.2
File Edit Simulation Flowsheet PFD Tools Window Help
Environment: Case
Mode: Stea
PFD - Case (Main)
1
Optional Info : 1 -- Unknown Temperature
Optional Info : 1 -- Unknown Pressure
Case (Main)
FLOW SHEET
CUSTOM COLUMN
MB

Figure 1-9

Specifying Material Streams

To enter information about the material stream, double click on the stream to show the window shown in Figure 1-10. It is within this window that the user specifies the details regarding the material stream. For material stream that will be used as an input, we need to specify **four** variables. Within HYSYS environment, input material stream always have four degree of freedoms. Meaning, we need to supply four information in order to fulfill the requirement for HYSYS to start its calculations.

> ***Tips:*** *Four variables needed for input stream are composition, flowrate, and two from temperature, pressure or vapor/phase fraction.*

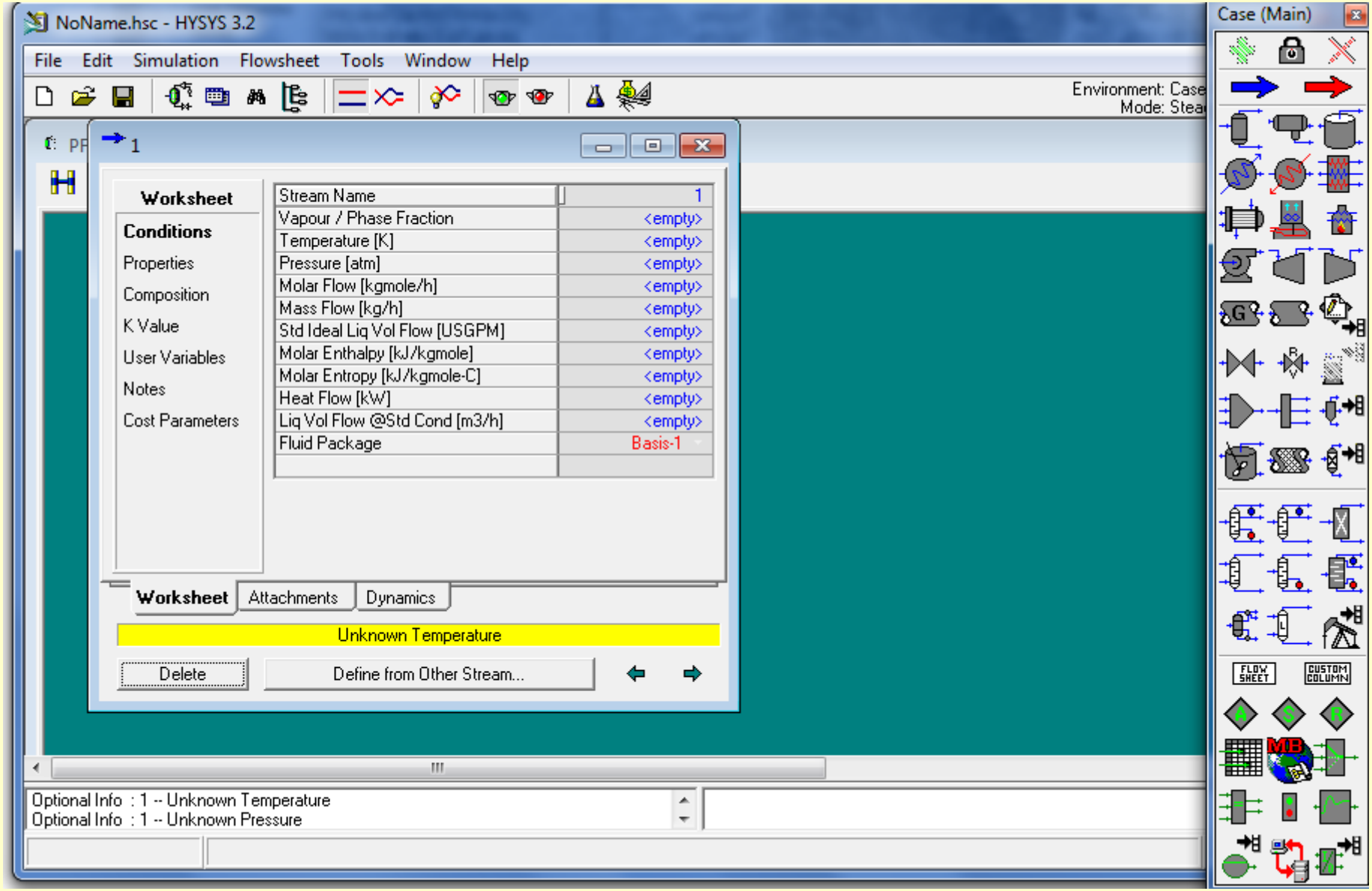
NoName.hsc - HYSYS 3.2
File Edit Simulation Flowsheet Tools Window Help
Environment: Case
Mode: Stea
Case (Main)
1
Worksheet
Conditions
Properties
Composition
K Value
User Variables
Notes
Cost Parameters
Stream Name 1
Vapour / Phase Fraction <empty>
Temperature [K] <empty>
Pressure [atm] <empty>
Molar Flow [kgmole/h] <empty>
Mass Flow [kg/h] <empty>
Std Ideal Liq Vol Flow [USGPM] <empty>
Molar Enthalpy [kJ/kgmole] <empty>
Molar Entropy [kJ/kgmole-C] <empty>
Heat Flow [kW] <empty>
Liq Vol Flow @Std Cond [m3/h] <empty>
Fluid Package Basis-1
Worksheet Attachments Dynamics
Unknown Temperature
Delete
Define from Other Stream...
Optional Info : 1 -- Unknown Temperature
Optional Info : 1 -- Unknown Pressure

Figure 1-10

From Figure 1-10, you will see the warning yellow message bar at the bottom of the window indicating what information is needed (unknown compositions). Just follow what the message wants, for example, the first thing that you need to supply is compositions. In order to specify

the composition of the stream, select the "Composition" option from this list to display the window in Figure 1-11. It is within this window that the user specifies the composition of the stream. Note that only the components that you specified in the simulation basis manager will appear in this list. You can specify the composition in many different ways by clicking on the "Basis..." button. The HYSYS default is mole fractions, however the user can also specify mass fractions, liquid volume fractions, or flows of each component. If the user is specifying fractions, all fractions must add up to 1. Enter mole fraction of **1** in the H2O section to indicate 1 mole fraction of water.

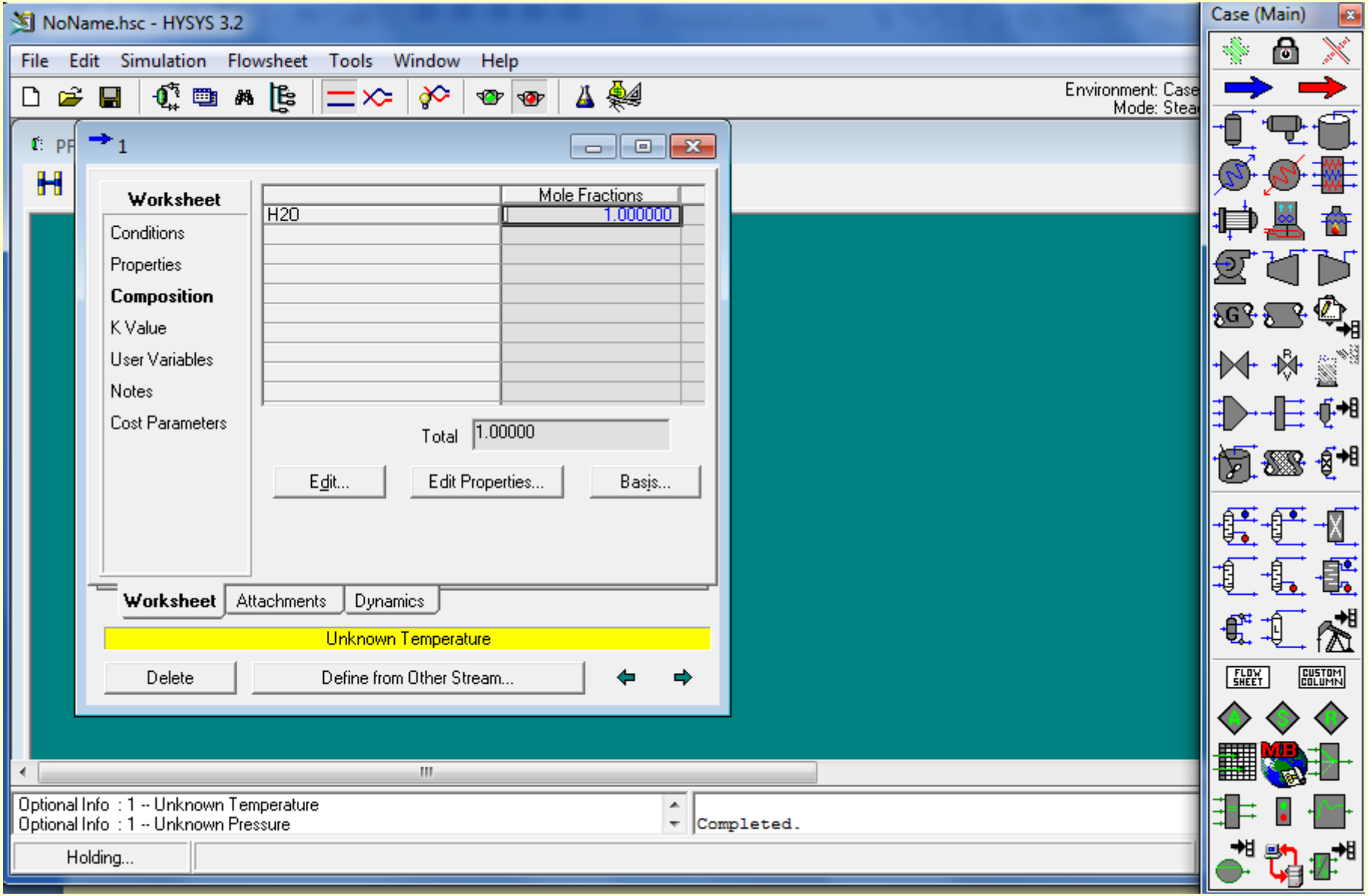
NoName.hsc - HYSYS 3.2
File Edit Simulation Flowsheet Tools Window Help
Environment: Case
Mode: Stea
Case (Main)
1
Worksheet
Conditions
Properties
Composition
K Value
User Variables
Notes
Cost Parameters
Mole Fractions
H2O
1.000000
Total
1.00000
Edit...
Edit Properties...
Basis...
Worksheet
Attachments
Dynamics
Unknown Temperature
Delete
Define from Other Stream...
FLOW SHEET
CUSTOM COLUMN
Optional Info : 1 -- Unknown Temperature
Optional Info : 1 -- Unknown Pressure
Completed.
Holding...

Figure 1-11

Next, the warning yellow message bar indicates that you need to specify the input temperature for this stream. In order to specify the temperature of the stream, select the "Conditions" option from this list to display the window in Figure 1-12. It is within this window that the user specifies the temperature of the stream.

When entering the conditions for a stream, it is not necessary to enter the values in the default units provided. When the user begins to enter a value in one of the cells, a drop down arrow appears in the units box next to the cell. By clicking on this drop down arrow, the user can specify any unit for the corresponding value and HYSYS will automatically convert the value to the default unit set. Enter the temperature of **25** in the temperature section to indicate the temperature of 25°C. Next, the yellow warning message bar indicates that you need to specify the input pressure for this stream. In the same window, enter the pressure of **1** in the pressure section to indicate the pressure of 1 bar as shown in Figure 1-13.

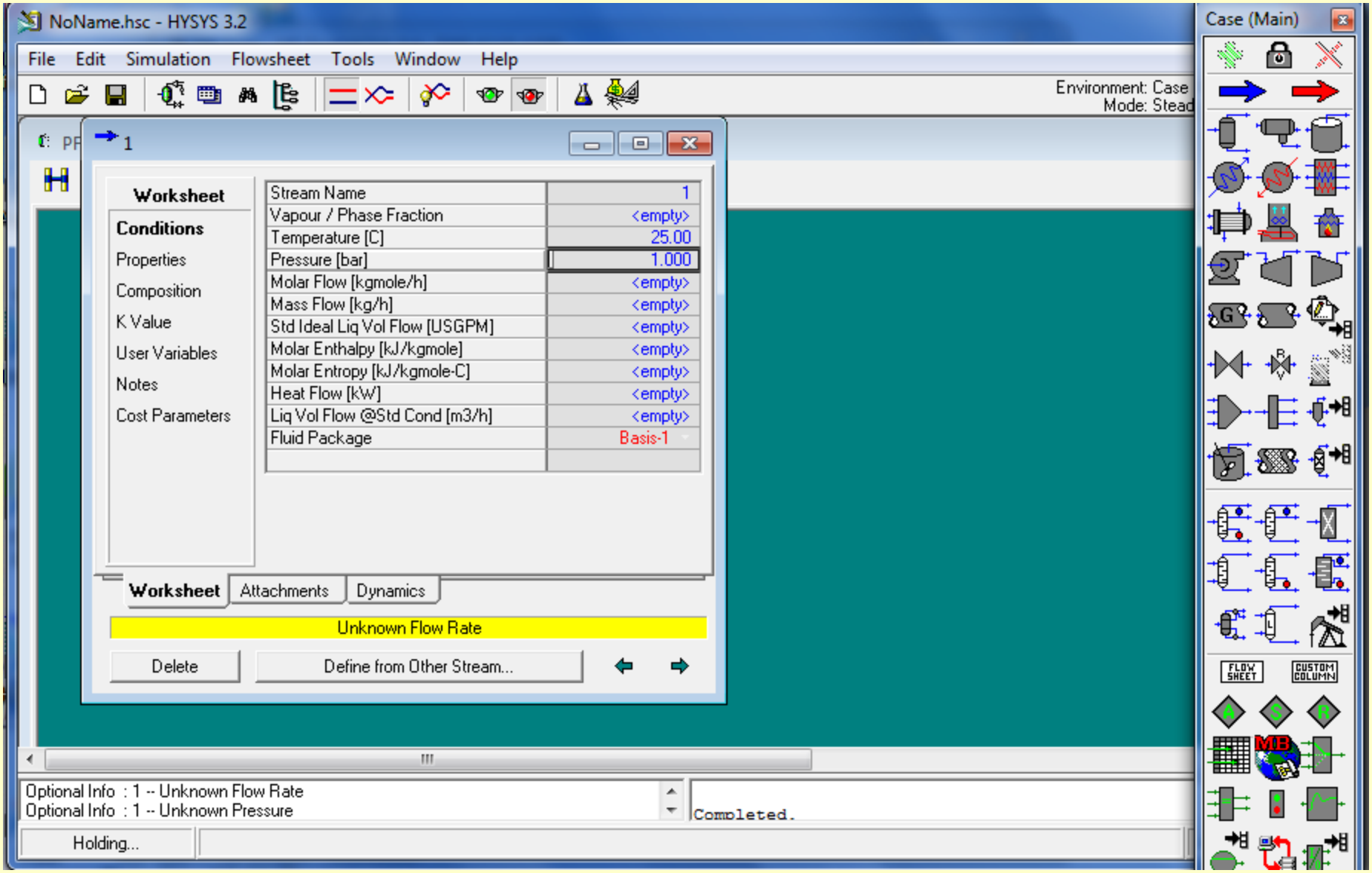
NoName.hsc - HYSYS 3.2
File Edit Simulation Flowsheet Tools Window Help
Environment: Case
Mode: Stead
Case (Main)
1
Worksheet
Conditions
Properties
Composition
K Value
User Variables
Notes
Cost Parameters
Stream Name 1
Vapour / Phase Fraction <empty>
Temperature [C] 25.00
Pressure [bar] 1.000
Molar Flow [kgmole/h] <empty>
Mass Flow [kg/h] <empty>
Std Ideal Liq Vol Flow [USGPM] <empty>
Molar Enthalpy [kJ/kgmole] <empty>
Molar Entropy [kJ/kgmole-C] <empty>
Heat Flow [kW] <empty>
Liq Vol Flow @Std Cond [m3/h] <empty>
Fluid Package Basis-1
Worksheet Attachments Dynamics
Unknown Flow Rate
Delete
Define from Other Stream...
FLOW SHEET
CUSTOM COLUMN
Optional Info : 1 -- Unknown Flow Rate
Optional Info : 1 -- Unknown Pressure
Completed.
Holding...

Figure 1-12

Next, the last variable that you need to specify is flowrate. For this, you have two options either to specify molar flowrate or mass flowrate. In the same window, enter the molar flowrate of **100** in the molar flowrate section to indicate the flowrate of 100 kgmole/h as shown in Figure 1-14.

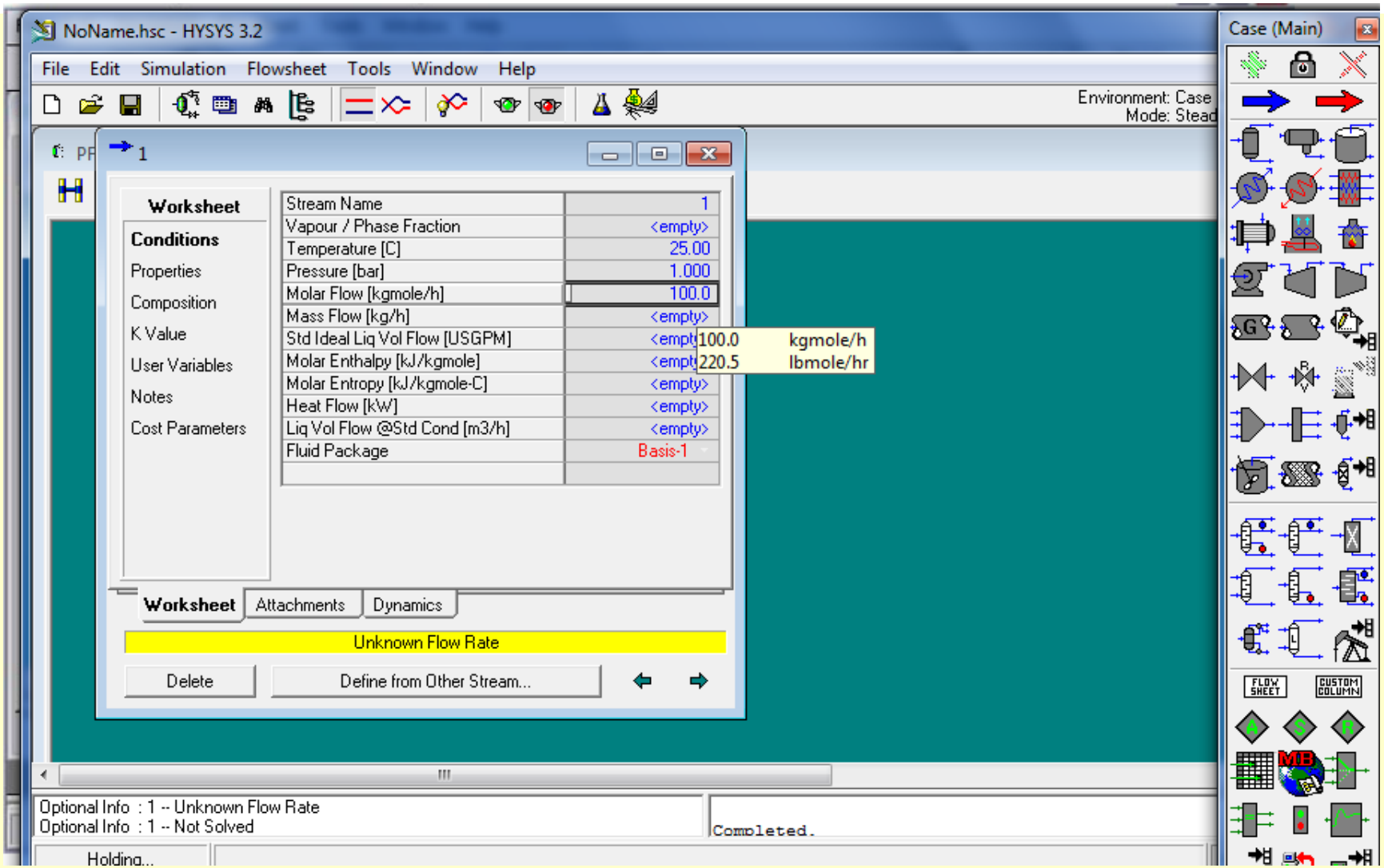

Figure 1-13