What is Aspen?
7 Basic Steps

• 1 – Setting your session preferences
• 2 – Building the simulation
• 3 – Entering the simulation environment
• 4 – Using the workbook
• 5 – Installing Unit Operations
• 6 – Run Your Simulation
• 7 – Viewing and analyzing the results
Building Tools

- **PFD**, Process Flow Diagram
- **Workbook**, The Workbook displays information about streams and unit operations in a tabular format, while the PFD is a graphical representation of the flow sheet.
Setting your session preferences

Start Aspen Hysys and create a new case. From the tools menu, select preferences. The session preferences property view appears. The simulation Tab, option page should be visible. Ensure that the Use Model Property Views checkbox is clear.
Creating a new unit set

Click the Variables tab, then select the Units page. The first task you perform when building the simulation case is choosing a unit set. Aspen HYSYS does not allow you to change any of the three default unit sets listed, however, you can create a new unit set by cloning an existing one.
Building the Simulation

1. Click the New Case icon. The Simulation Basis Manager appears.
2. Create a Fluid Package. A Fluid Package, at minimum, contains the components and property method that Aspen HYSYS will use in its calculations for a particular flow sheet.
Selecting Components

On the Components tab of the Simulation Basis Manager property view, click the Add button in the Component Lists group. The Component List property view appears.
Selecting components

Each component can appear in three forms, corresponding to the three radio buttons that appear above the component list. When you add a component, it then appears in the Selected Components list.

To view the properties of one or more components, select the component(s) and click the View Component button. Aspen HYSYS opens the property view(s) for the component(s) you select.
Creating a Fluid Package

1. Click the Fluid Pkgs tab of the Simulation Basis Manager.
2. Click the Add button. The Fluid Package property view appears.
Click the Fluid Pkgs tab to view a summary of the completed Fluid Package. The list of Current Fluid Packages displays the new Fluid Package, Basis-1, showing the number of components (NC) and property package (PP).
Entering the Simulation Environment

To leave the Basis environment and enter the Simulation environment, do one of the following:

- Click the **Enter Simulation Environment button on the** Simulation Basis Manager.
- Click the **Enter Simulation Environment icon on the** toolbar.
## Simulation Environment

<table>
<thead>
<tr>
<th>Features</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Workbook</strong></td>
<td>A multiple-tab property view containing information about the objects (streams and unit operations) in the simulation case.</td>
</tr>
<tr>
<td></td>
<td>By default, the Workbook has four tabs, namely <strong>Material Streams</strong>, <strong>Compositions</strong>, <strong>Energy Streams</strong>, and <strong>Unit Ops</strong>. You can edit the Workbook by adding or deleting tabs and changing the information displayed on any tab.</td>
</tr>
<tr>
<td><strong>Object Palette</strong></td>
<td>A floating palette of buttons that can be used to add streams and unit operations. You can toggle the palette open or closed by pressing F4, or by choosing Open/Close Object Palette from the Flowsheet menu.</td>
</tr>
</tbody>
</table>
Installing the feed Streams

you will install feed streams using either the Workbook or using object palette.

1. Click the Workbook icon on the toolbar to make the Workbook active.
2. On the Material Streams tab, click in the **New** cell in the Name row.
3. Type the new stream name, then press ENTER Aspen HYSYS automatically creates the new stream.

Next you will define the feed conditions for temperature and pressure and flow.
Installing the feed Stream
Providing Compositional Input

Now that the stream conditions have been specified, your next task is to input the composition.

1. In the Workbook, double-click the **Molar Flow cell**.

The Input Composition for Stream property view appears. This property view allows you to complete the compositional input.
Installing the Feed Stream

There is an alternative method for adding a stream. Double-click the **Material Stream icon on the Object Palette.**

When you create the new stream, the stream’s property view also appears, displaying the **Conditions page of the Worksheet tab**
Installing the Feed stream
Installing Unit operation (Mixer)

Simply add the Unit Operation by Double-clicking the unit operation from the object Palette.
Installing Unit operation (Mixer)

The property view for the Mixer appears. The unit operation property view contains all the information required to define the operation, organized into tabs and pages. The Design, Rating, Worksheet, and Dynamics tabs.
Installing Unit operation

For the Mixer you just need to specify the feed streams (Multiple streams) and outlet stream(one stream) by clicking in the related cells and typing a name for the streams, even if this stream does not exist Aspen creates that stream.

The status indicator displays a green **OK and is green, indicating that the operation and attached streams are completely calculated.**

The Connections page is now complete.
Installing Unit operation (Mixer)

Click the **Parameters page**. In the Automatic Pressure Assignment group, keep the default setting of **Set Outlet to Lowest Inlet**.

Aspen HYSYS calculates the outlet stream by combining the two inlets and flashing the mixture at the lowest pressure of the inlet streams.
Installing Unit Operation (Splitter/Tee)

- Splitter or Tee splits the feed stream into 2 or 3 or more. Add Splitter to PFD by double clicking the Tee Icon in object Palette.
Installing Unit Operation (Splitter/Tee)

- Aspen Needs some more info to solve the splitter. That would be Flow Ratios of the output stream. In the parameters page of the Design Tab specify the flow ratios.
Calculation Status

Aspen HYSYS uses colour-coding to indicate calculation status for objects, both in the object property views, and in the flowsheet. If you recall, the status bar indicator at the bottom of a property view for a stream or operation indicates the current state of the object.

The following table lists and describes the three colour status:

<table>
<thead>
<tr>
<th>Indicator Status</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Red Status</td>
<td>A major piece of defining information is missing from the object. For example, a feed or product stream is not attached to a Separator. The status indicator is red and an appropriate warning message is displayed.</td>
</tr>
<tr>
<td>Yellow Status</td>
<td>All major defining information is present, but the stream or operation has not been solved because one or more degrees of freedom is present. For example, a Cooler whose outlet stream temperature is unknown. The status indicator is yellow and an appropriate warning message is displayed.</td>
</tr>
<tr>
<td>Green Status</td>
<td>The stream or operation is completely defined and solved. The status indicator is green and an OK message is displayed.</td>
</tr>
</tbody>
</table>
Another colour scheme is used to indicate the status of streams. For material streams, a \textit{dark blue icon} indicates the stream has been flashed and is entirely known. A \textit{light blue icon} indicates the stream cannot be flashed until some additional information is supplied. Similarly, a \textit{dark red icon} is for an energy stream with a known duty, while a \textit{purple icon} indicates an unknown duty.
Viewing the Results

Click the **Workbook icon** to access the calculated results for the Main Flowsheet.