HYSYS®

Upstream Option Guide

Copyright

November 2004

Aspen ACOL™, Aspen Adsim®, Aspen Advisor™, Aspen Aerotran®, Aspen Alarm & Event™, Aspen APLE™, Aspen Apollo Desktop™, Aspen Apollo Online™, Aspen AssetBuilder™, Aspen ATOMS™, Aspen Automated Stock Replenishment™, Aspen Batch Plus®, Aspen Batch.21™, Aspen BatchCAD™, Aspen BatchSep™, Aspen Calc™, Aspen Capable-to-Promise®, Aspen CatRef®, Aspen Chromatography®, Aspen Cim-IO for ACS™, Aspen Cim-IO for Csi VXL™, Aspen Cim-IO for Dow MIF™, Aspen Cim-IO for G2™, Aspen Cim-IO for GSE D/3™, Aspen Cim-IO for Hewlett-Packard RTAP™, Aspen Cim-IO for Hitachi PLC (H04E)™, Aspen Cim-IO for Intellution Fix™, Aspen Cim-IO for Melsec™, Aspen Cim-IO for WonderWare InTouch™, Aspen Cim-IO for Yokogawa Centum CS™, Aspen Cim-IO for Yokogawa Centum XL™, Aspen Cim-IO for Yokogawa EW3™, Aspen Cim-IO Interfaces™, Aspen Cim-IO Monitor™, Aspen Cim-IO™, Aspen Collaborative Demand Management™, Aspen Collaborative Forecasting™, Aspen Compliance.21™, Aspen COMThermo TRC Database™, Aspen COMThermo®, Aspen Cost Factor Manual™, Aspen Crude Manager™, Aspen Crude Margin Evaluation™, Aspen Custom Modeler®, Aspen Data Source Architecture™, Aspen Decision Analyzer™, Aspen Demand Manager™, Aspen DISTIL™, Aspen Distribution Scheduler™, Aspen DMCplus® Composite, Aspen DMCplus® Desktop, Aspen DMCplus® Online, Aspen DPO™, Aspen Dynamics®, Aspen eBRS™, Aspen Enterprise Model™, Aspen ERP Connect™, Aspen FCC®, Aspen FIHR™, Aspen FLARENET™, Aspen Fleet Operations Management™, Aspen Framework™, Aspen FRAN™, Aspen Fuel Gas Optimizer Desktop™, Aspen Fuel Gas Optimizer Online™, Aspen General Construction Standards™, Aspen Hetran®, Aspen HX-Net®, Aspen Hydrocracker®, Aspen Hydrotreater™, Aspen HYSYS Amines™, Aspen HYSYS Crude™, Aspen HYSYS Dynamics™, Aspen HYSYS OLGAS 3-Phase™, Aspen HYSYS OLGAS™, Aspen HYSYS OLI Interface™, Aspen HYSYS Tacite™, Aspen HYSYS Upstream Dynamics™, Aspen HYSYS Upstream™, Aspen HYSYS®, Aspen Icarus Process Evaluator®, Aspen Icarus Project Manager®, Aspen InfoPlus.21®, Aspen Inventory Balancing™, Aspen IQ Desktop™, Aspen IQ Online™, Aspen IQmodel Powertools™, Aspen Kbase®, Aspen LIMS Interface™, Aspen Local Security™, Aspen LPIMS™, Aspen MBO™, Aspen MIMI®, Aspen MPIMS™, Aspen Multivariate Server™, Aspen MUSE™, Aspen NPIMS™, Aspen OnLine®, Aspen Operations Manager - Event Management™, Aspen Operations Manager - Integration Infrastructure™, Aspen Operations Manager - Peformance Scorecarding™, Aspen Operations Manager - Role Based Visualization™, Aspen Order Credit Management™, Aspen Orion Planning™, Aspen Orion™, Aspen PEP Process Library™, Aspen PIMS Blend Model Library™, Aspen PIMS Distributed Processing™, Aspen PIMS Enterprise Edition™, Aspen PIMS Mixed Integer Programming™, Aspen PIMS Simulator Interface™, Aspen PIMS Solution Ranging™, Aspen PIMS Submodel Calculator™, Aspen PIMS XNLP Optimizer™, Aspen PIMS™, Aspen PIPESYS™, Aspen PIPE™, Aspen Planning Accuracy™, Aspen Plant Planner & Scheduler™, Aspen Plant Scheduler Lite™, Aspen Plant Scheduler™, Aspen Plus OLI Interface™, Aspen Plus®, Aspen Polymers Plus®, Aspen PPIMS™, Aspen Process Data™, Aspen Process Explorer™, Aspen Process Manual™, Aspen Process Order™, Aspen Process Plant Construction Standards™, Aspen Process Recipe®, Aspen Process Tools™, Aspen Product Margin & Blending Evaluation™, Aspen Production Control Web Server™, Aspen ProFES® 2P Tran, Aspen ProFES® 2P Wax, Aspen ProFES® 3P Tran, Aspen ProFES® Tranflo, Aspen Properties®, Aspen Pumper Log™, Aspen Q Server™, Aspen RateSep™, Aspen RefSYS CatCracker™, Aspen RefSYS Spiral™, Aspen RefSYS™, Aspen Report Writer™, Aspen Resource Scheduling Optimization™, Aspen RTO Watch Cim-IO Server™, Aspen RTO Watch Server™, Aspen Scheduling & Inventory Management™, Aspen SmartStep Desktop™, Aspen SmartStep Online™, Aspen SQLplus™, Aspen Supply Chain Analytics™, Aspen Supply Chain Connect™, Aspen Supply Planner™, Aspen Tank Management™, Aspen TASC-Mechanical™, Aspen TASC™, Aspen Teams®, Aspen Terminals Management™, Aspen TICP™, Aspen Transition Manager™, Aspen Turbo PIMS™, Aspen Utilities™, Aspen Voice Fulfillment Management™, Aspen Watch Desktop™, Aspen Watch Server™, Aspen Water™, Aspen Web Fulfillment Management™, Aspen WinRace Database™, Aspen XPIMS™, Aspen Zyqad Development Version™, Aspen Zyqad™, SLM™, SLM Commute™, SLM Config Wizard™, the aspen leaf logo, and Plantelligence are trademarks or registered trademarks of Aspen Technology, Inc., Cambridge, MA.

All other brand and product names are trademarks or registered trademarks of their respective companies.

This manual is intended as a guide to using AspenTech's software. This documentation contains AspenTech proprietary and confidential information and may not be disclosed, used, or copied without the prior consent of AspenTech or as set forth in the applicable license agreement. Users are solely responsible for the proper use of the software and the application of the results obtained.

Although AspenTech has tested the software and reviewed the documentation, the sole warranty for the software may be found in the applicable license agreement between AspenTech and the user. **ASPENTECH MAKES NO WARRANTY OR REPRESENTATION, EITHER EXPRESSED OR IMPLIED, WITH RESPECT TO THIS DOCUMENTATION, ITS QUALITY, PERFORMANCE, MERCHANTABILITY, OR FITNESS FOR A PARTICULAR PURPOSE**.

Corporate

Aspen Technology, Inc.	Phone: (617) 949-1000
Ten Canal Park	Toll Free: (1) (888) 996-7001
Cambridge, MA 02141-2201	Fax: (617) 949-1030
USA	Website http://www.aspentech.com

Table of Contents

1 Black Oil

1.1 Black Oil Tutorial Introduction

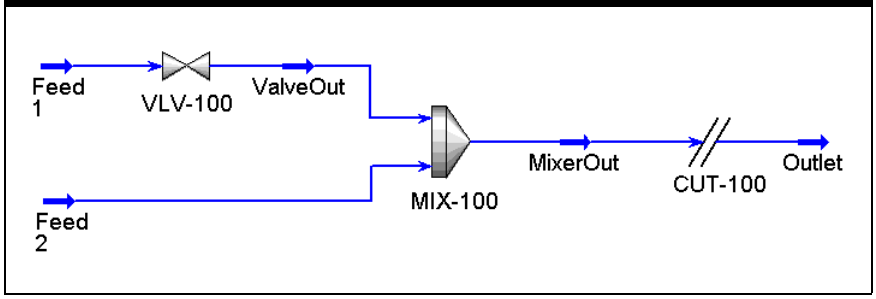
In today's oil and gas industry, it becomes increasingly necessary to use a compositional model and equation of state to accurately model the behaviour of a petroleum fluid downstream of the well and flowlines. The HYSYS Upstream Option translates from black oil to a compositional model using the gas composition and HYSYS oil characterization.

In HYSYS, Black Oil describes a class of phase behaviour and transport property models. Black oil correlations are typically used when a limited amount of oil and gas information is available in the system. Oil and gas fluid properties are calculated from correlations with their respective specific gravity (as well as a few other easily measured parameters).

Black Oil is not typically used for systems that would be characterized as gas-condensate or dry gas, but rather for systems where the liquid phase is a non-volatile oil (and consequently there is no evolution of gas, except for that which is dissolved in the oil).

In this Tutorial, two black oil streams at different conditions and compositions are passed through a mixer to blend into one black oil stream. The blended black oil stream is then fed to the Black Oil Translator where the blended black oil stream data is transitioned to a HYSYS material stream. A flowsheet for this process is shown below.

Figure 1.1



The following pages will guide you through building a HYSYS case for modeling this process. This example will illustrate the complete construction of the simulation, from selecting the property package and components, to installing streams and unit operations, through to examining the final results. The tools available in the HYSYS interface will be used to illustrate the flexibility available to you.

The simulation will be built using these basic steps:

1. Create a unit set and set the Black Oil default options.
2. Select the components.
3. Add a Neotec Black Oil property package.
4. Create and specify the feed streams.
5. Install and define the unit operations prior to the translator.
6. Install and define the translator.
7. Add a Peng-Robinson property package.

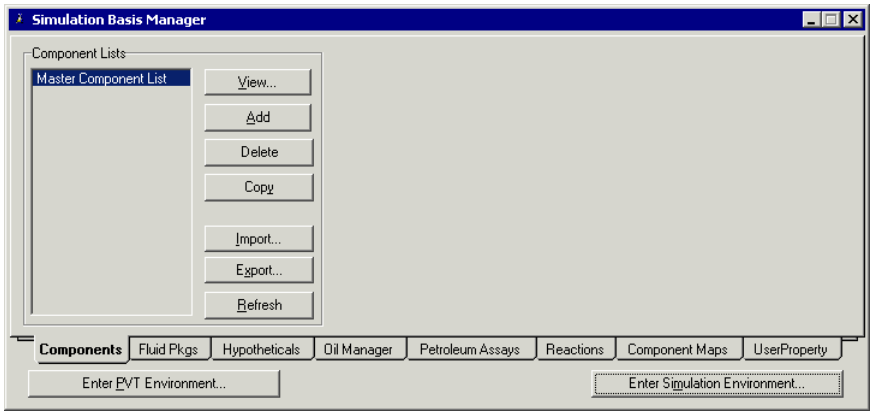
1.2 Setting the Session Preferences

New Case icon

1. To start a new simulation case, do **one** of the following:
   - From the **File** menu, select **New** and then **Case**.
   - Click the **New Case** icon.

   The Simulation Basis Manager appears:

Figure 1.2

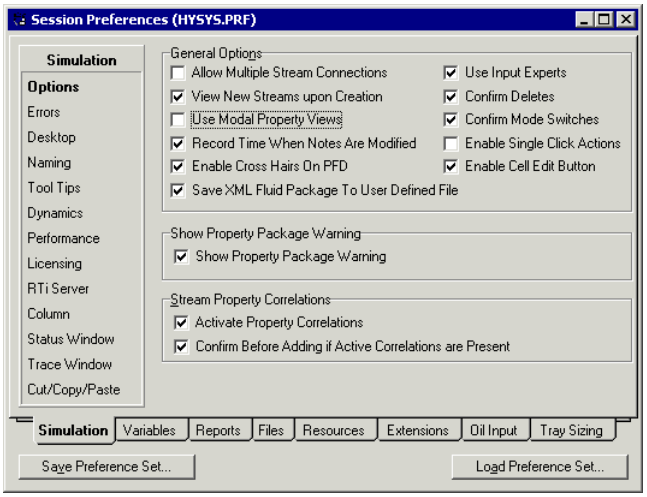


Next you will set your Session Preferences before building a case.

2. From the **Tools** menu, select **Preferences**.

The Session Preferences view appears. You should be on the **Options** page of the **Simulation** tab.

Figure 1.3



3. In the General Options group, ensure the **Use Modal Property Views** checkbox is unchecked so that you can access multiple views at the same time.

1.2.1 Creating a New Unit Set

The first step in building the simulation case is choosing a unit set. Since HYSYS does not allow you to change any of the three default unit sets listed (in other words, EuroSI, Field, and SI), you will create a new unit set by cloning an existing one. For this example, a new unit set will be made based on the HYSYS Field set, which you will then customize.

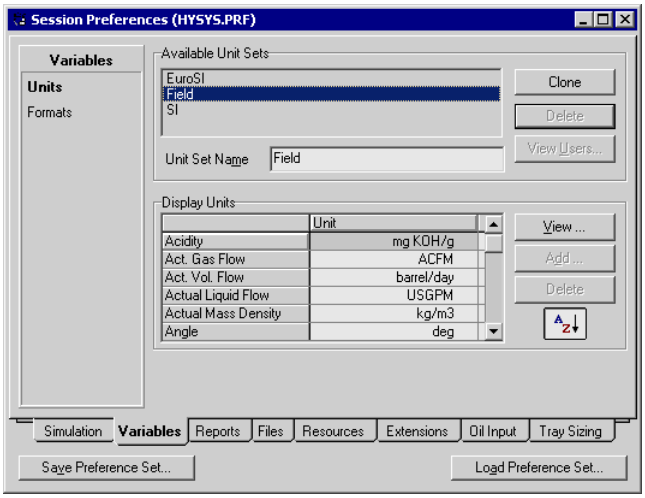
To create a new unit set, do the following:

1. In the Session Preferences view, click the **Variables** tab.
2. Select the **Units** page if it is not already selected.

The default Preference file is named **hysys.PRF**. When you modify any of the preferences, you can save the changes in a new Preference file by clicking the **Save Preference Set** button. HYSYS prompts you to provide a name for the new Preference file, which you can load into any simulation case by clicking the **Load Preference Set** button.

3. In the Available Unit Sets group, highlight **Field** to make it the active set.

Figure 1.4

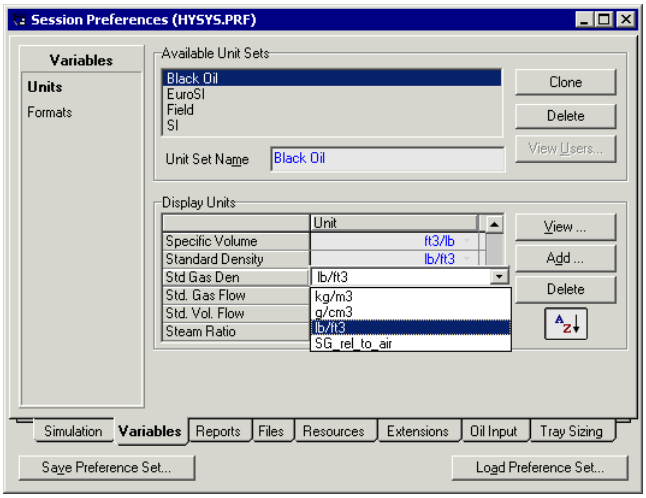


4. Click the **Clone** button. A new unit set named **NewUser** appears. This unit set becomes the currently Available Unit Set.
5. In the **Unit Set Name** field, rename the new unit set as **Black Oil**. You can now change the units for any variable associated with this new unit set.

   In the Display Units group, the current default unit for **Std Gas Den** is lb/ft3. In this example we will change the unit to SG_rel_to_air.
6. Scroll through the table in the Display Units group, until you find the **Std Gas Den** variable.

7. To view the available units for **Std Gas Den**, click the drop-down arrow in the cell beside the **Std Gas Den** cell.

Figure 1.5



8. From the drop-down list, select **SG_rel_to_air**.
9. Repeat the previous three steps to change the **Standard Density** unit to **SG_60/60 api**.

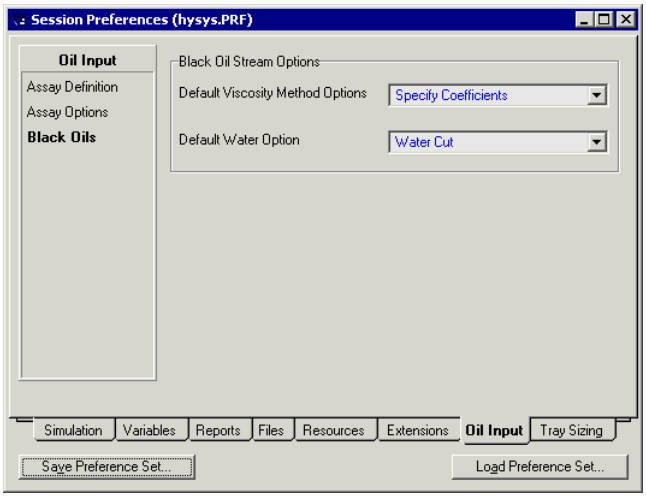
   Your Black Oil unit set is now defined.

1.2.2 Setting Black Oil Stream Default Options

To set the Black Oil stream default options:

1. Click on the **Oil Input** tab in the Session Preference view.
2. In the Session Preferences view, select the **Black Oils** page.

Figure 1.6



In the Black Oil Stream Options group, you can select the methods for calculating the viscosity, and displaying the water content for **all** the black oil streams in your simulation. For now you will leave the settings as default.

Close icon

3. Click the **Close** icon (in the top right corner) to close the Session Preferences view. You will now add the components and fluid package to the simulation.

1.3 Setting the Simulation Basis

The Simulation Basis Manager allows you to create, modify, and manipulate fluid packages in your simulation case. As a minimum, a Fluid Package contains the components and property method (for example, an Equation of State) HYSYS will use in its calculations for a particular flowsheet. Depending on what is required in a specific flowsheet, a Fluid Package may also contain other information such as reactions and interaction parameters. You will first define your fluid package by selecting the components in this simulation case.

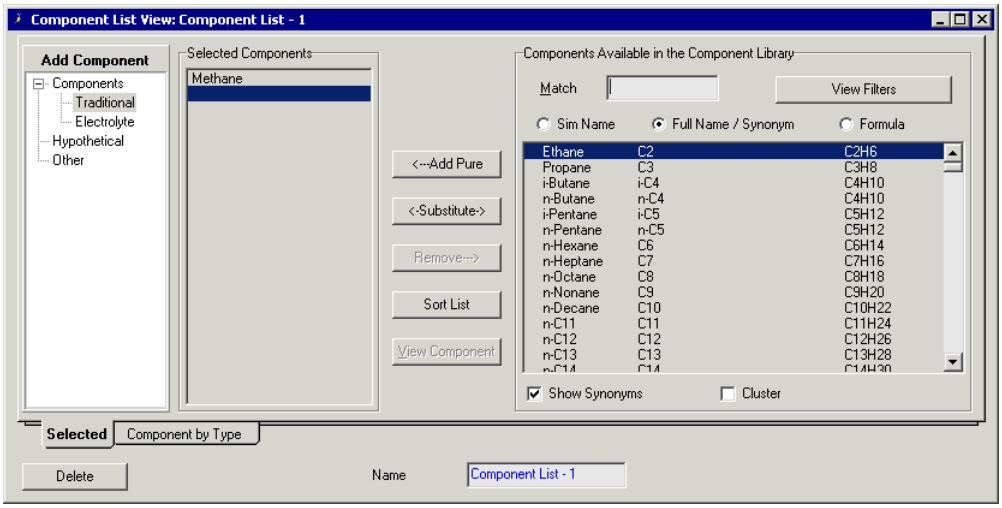
1.3.1 Selecting Components

HYSYS has an internal stipulation that at least one component must be added to a component list that is associated to a fluid package. To fulfil this requirement you must add a minimum of a single component even when the compositional data is not needed. For black oil streams, depending on the information available, you have the option to either specify the gas components compositions or the gas density to define the gas phase of the stream.

To add components to your simulation case:

1. Click on the **Components** tab in the Simulation Basis Manager.
2. Click the **Add** button. The Component List view is displayed.

Figure 1.7



For more information on adding and viewing components, refer to **Chapter 1 - Components** in the **HYSYS Simulation Basis** guide.

3. In this tutorial, add the following components: **C1**, **C2**, **C3**, **i-C4**, **n-C4**, **i-C5**, **n-C5**, and **C6**.
4. Close the Component List View to return to the Simulation Basis Manager view.

   If the Simulation Basis Manager is not visible, click the **Home View** icon from the tool bar.

1.3.2 Creating a Fluid Package

In this tutorial, since a Black Oil Translator is used in transitioning a Black Oil stream to a HYSYS compositional stream, two property packages are required in the simulation. You will first add the Neotec Black Oil property package and later in the tutorial after, you have installed the black oil translator, you will add the Peng-Robinson property package.

Adding the Neotec Black Oil Property Package

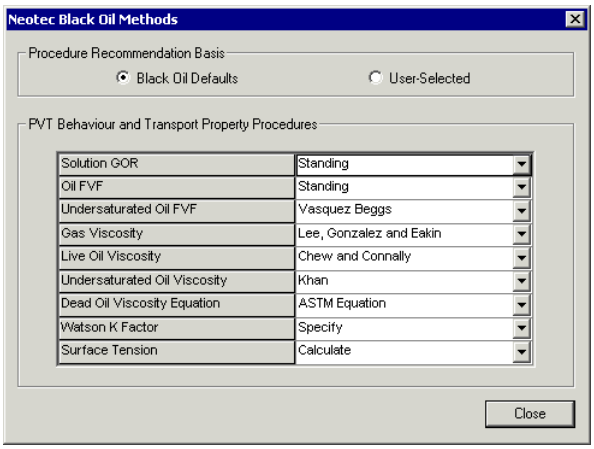
To add the Neotec Black Oil Property Package to your simulation:

1. From Simulation Basis Manager, click the **Fluid Pkgs** tab.
2. Click the **Add** button in the Current Fluid Packages group. The Fluid Package Manager appears.
3. In the Component List Selection group, select **Component List - 1** from the drop-down list.
4. From the list of available property packages in the Property Package Selection group, select **Neotec Black Oil**.

   The Neotec Black Oil Methods view appears.

You can also filter the list of available property packages by clicking the **Miscellaneous Type** radio button in the Property Package Filter group. From the filtered list you can select **Neotec Black Oil**.

Figure 1.8

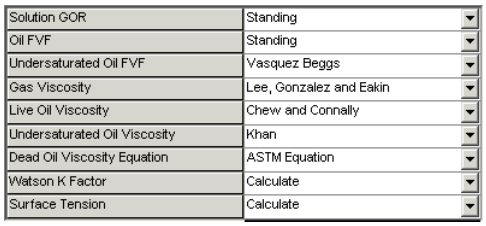


Refer to **Appendix A - Neotec Black Oil Methods** for more information on the black oil methods available and other terminology.

The Neotec Black Oil Methods view displays the nine PVT behaviour and transport property procedures, and each of their calculation methods.

5. In this tutorial, you want to have the Watson K Factor calculated by the simulation. The default option for the **Watson K Factor** is set at **Specify**. Thus, you will change the option to **Calculate** from the Watson K Factor drop-down list, as shown below.

Figure 1.9



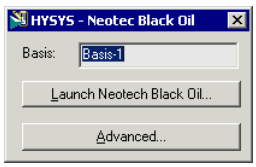
You can restore the default settings by clicking on the **Black Oil Defaults** radio button.

The User-Selected radio button is automatically activated when you select a Black Oil method that is not the default.

6. Click the **Close** button to close the Neotec Black Oil Methods view.

   The HYSYS Neotec Black Oil view appears.

Figure 1.10



7. In the **Basis** field, type in the new name **Black Oil** for the fluid package.
8. Close the HYSYS Neotec Black oil view by clicking the **Close** icon.

Close icon

The Black Oil fluid package is now completely defined. If you click on the **Fluid Pkgs** tab in the Simulation Basis Manger you can see that the list of Current Fluid Packages now

displays the Black Oil Fluid Package and shows the number of components (NC) and property package (PP). The newly created Black Oil Fluid Package is assigned by default to the main flowsheet. Now that the Simulation Basis is defined, you can install streams and operations in the Main Simulation environment.

9. 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 view.
   - Click the **Enter Simulation Environment** icon on the tool bar.


Enter Simulation Environment icon

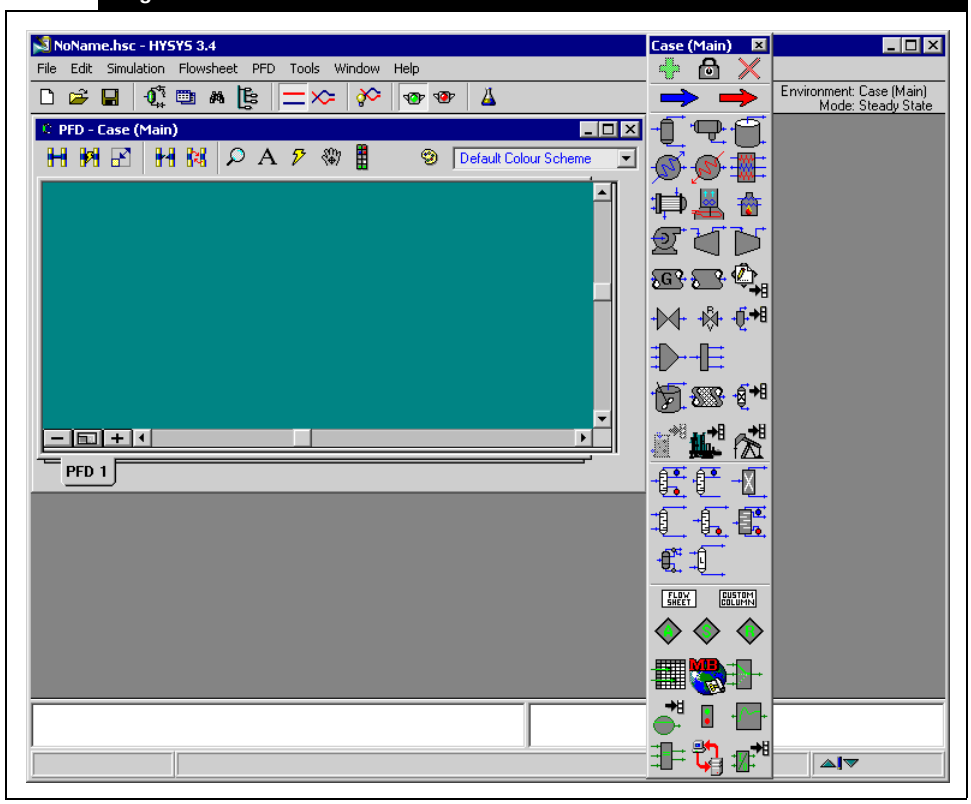
1.3.3 Entering the Simulation Environment

When you enter the Simulation environment, the initial view that appears depends on your current Session Preferences setting for the Initial Build Home View. Three initial views are available:

- PFD
- Workbook
- Summary

Any or all of these can be displayed at any time; however, when you first enter the Simulation environment, only one appears. In this example, the initial Home View is the PFD (HYSYS default setting).

Figure 1.11



There are several things to note about the Main Simulation environment. In the upper right corner, the Environment has changed from Basis to Case (Main). A number of new items are now available in the menu bar and tool bar, and the PFD and Object Palette are open on the Desktop.

The PFD and Object Palette are described below.

Objects	Description
PFD	The PFD is a graphical representation of the flowsheet topology for a simulation case. The PFD view shows operations and streams and the connections between the objects. You can also attach information tables or annotations to the PFD. By default, the view has a single tab. If required, you can add additional PFD pages to the view to focus in on the different areas of interest.
Object Palette	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 selecting the Open/Close Object Palette command from the Flowsheet menu.

You can also open the Object Palette by clicking the Object Palette icon in the PFD tool bar.

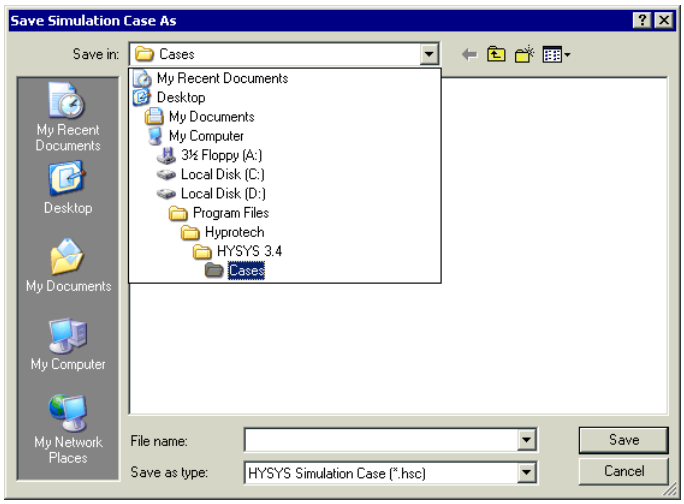
Before proceeding any further, save your case.

1. Do **one** of the following:
   - From the **File** menu, select **Save**.
   - Press **CTRL S**.
   - Click the **Save** icon on the tool bar.

   If this is the first time you have saved your case, the Save Simulation Case As view appears.

Save icon

Figure 1.12



When you choose to open an existing case by clicking the

Open Case icon , or by selecting **Open Case** from the File menu, a view similar to the one shown in **Figure 1.12** appears. The File Filter drop-down list will then allow you to retrieve backup (*.bk*) and HYSIM (*.sim) files in addition to standard HYSYS (*.hsc) files.

By default, the File Path is the Cases sub-directory in your HYSYS directory.

2. In the **File Name** cell, type a name for the case, for example **BlackOil**.

   You do not have to enter the ***.hsc** extension, HYSYS automatically adds it for you.

3. Once you have entered a file name, press the **ENTER** key or click the **Save** button.

   HYSYS saves the case under the name you have given it when you save in the future. The Save As view will not appear again unless you choose to give it a new name using the **Save As** command. If you enter a name that already exists in the current directory, HYSYS will ask you for confirmation before over-writing the existing file.

1.4 Building the Simulation

1.4.1 Installing the Black Oil Feed Streams

In this tutorial, you will install two black oil feed streams.

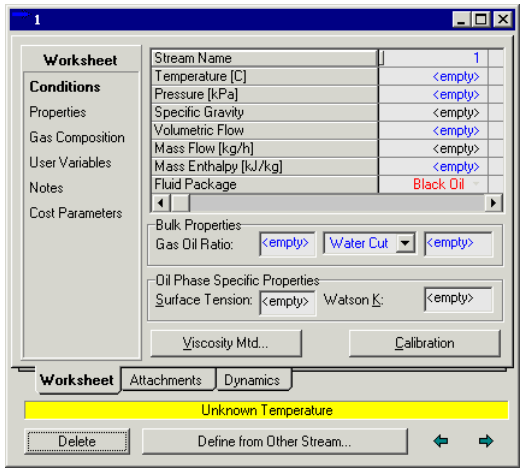
1. To add the first black oil stream to your simulation do one of the following:
   - From the **Flowsheet** menu, select **Add Stream**.
   - You can also add a new material stream by pressing the **F11** hot key.
   - From the **Flowsheet** menu, select **Palette**. The Object Palette appears.

     Double-click on the **Material Stream** icon.

Material Stream icon

The Black Oil Stream property view appears.

Figure 1.13



HYSYS displays three different phases in a black oil stream. The three phases are:

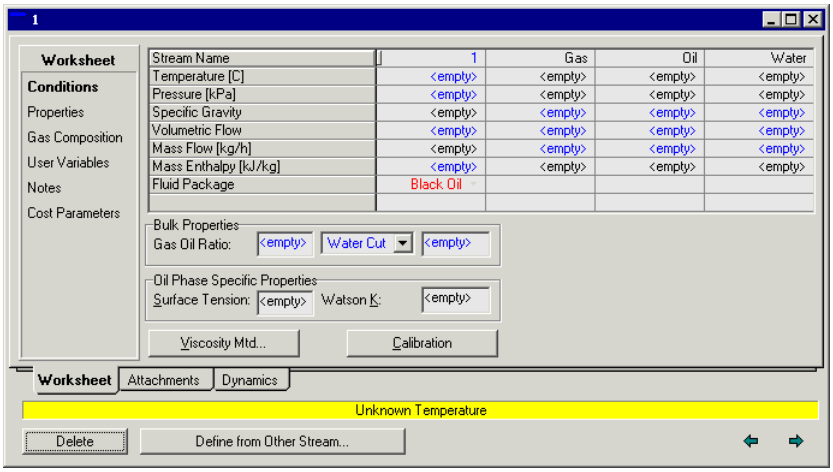
- Gas
- Oil
- Water

You can also use the horizontal scroll bar to view all the phase properties.

The first column is the overall stream properties column. You can view and edit the Gas, Oil, and Water phase properties by expanding the width of the default Black Oil stream property view.

The expanded stream property view is shown below.

Figure 1.14



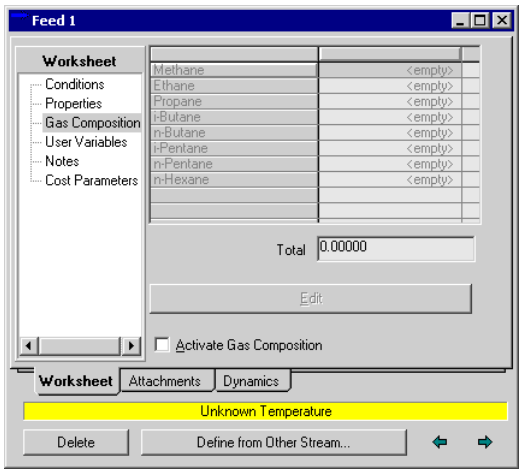
2. Rename the stream to **Feed 1** by typing the new stream name in the **Stream Name** cell of the Overall column (first column).

You can only rename the overall column, and that name appears on the PFD as the name for that black oil stream. You cannot change the phase name for the stream.

Next you will define the gas composition in **Feed 1**.

3. On the **Worksheet** tab, click on the **Gas Composition** page to begin the compositional input for the stream.

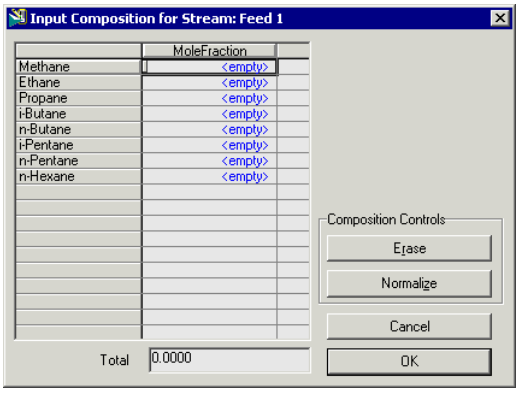
Figure 1.15



The Activate Gas Composition checkbox allows you to specify the compositions for each base component you selected in the Simulation Basis manager. After you have defined the gas composition for the black oil stream, HYSYS will automatically calculate the specific gravity for the gas phase. If gas composition information is not available, you can provide only the specific gas gravity on the Conditions page to define the black oil stream.

4. Check the **Activate Gas Composition** checkbox to activate the Gas Composition table.
5. Click on the **Edit** button. The Input Composition for Stream view appears. By default, you can only specify the stream compositions in mole fraction.

Figure 1.16

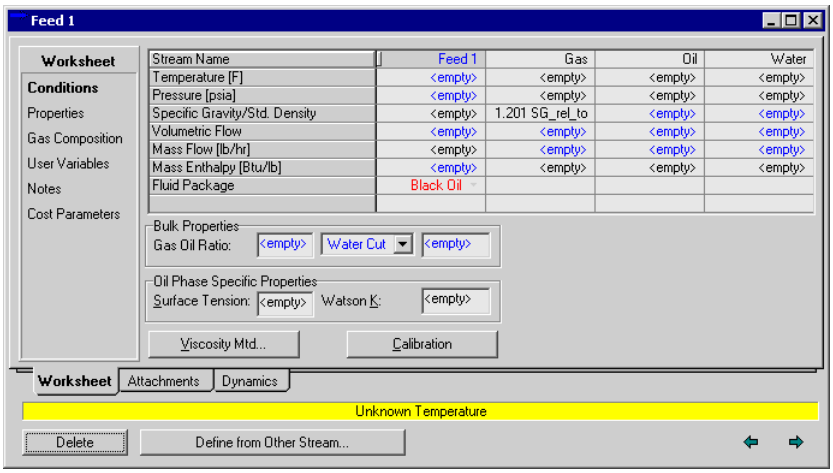


6. Enter the following composition for each component:

Component	Mole Fraction
Methane	0.3333
Ethane	0.2667
Propane	0.1333
i-Butane	0.2000
n-Butane	0.0677
i-Pentane	0.0000
n-Pentane	0.0000
n-Hexane	0.0000

7. Click the **Normalize** button to ensure that the mole fraction sum equals **1.0**.
8. Click the **OK** button, and HYSYS accepts the composition.
9. Click on the **Conditions** page on the **Worksheet** tab.

Figure 1.17



Next you will define the conditions for **Feed 1**.

10. In the overall column (first column), specify the following conditions:

In this cell...	Enter...
Temperature (°C)	50

In this cell...	Enter...
Pressure (kPa)	101.3
Volumetric Flow (barrel/day)	4500

HYSYS automatically assigns the same temperature and pressure to the Gas, Oil, and Water phases.

11. Specify the **Specific Gravity** for the Oil phase and Water phase to **0.847 SG_60/60 api** and **1.002 SG_60/60 api**, respectively.

    Next you will specify the bulk properties for **Feed 1**.

12. In the Bulk Properties group, specify a Gas Oil Ratio (**GOR**) of **1684 SCF/bbl**, and **Water Cut** of **15**%.

Figure 1.18

The Gas Oil Ratio is the ratio of the gas volumetric flow to oil volumetric flow at stock tank conditions. The Gas Oil Ratio will be automatically calculated if the volumetric flows of the gas, oil, and water phases are known. In this tutorial, the volumetric flowrates for the three phases are calculated by the Gas Oil Ratio and Water Cut.

The water content in the Black Oil stream can be expressed in two ways:

- **Water Cut**. The water cut is expressed as a percentage.

$$\text{Water Cut} = \frac{V_{water}}{V_{oil} + V_{water}} \tag{1.1}$$

where:

V_{water} = *volume of water*

V_{oil} = *volume of oil*

- **WOR**. A ratio of volume of water to the volume of oil.

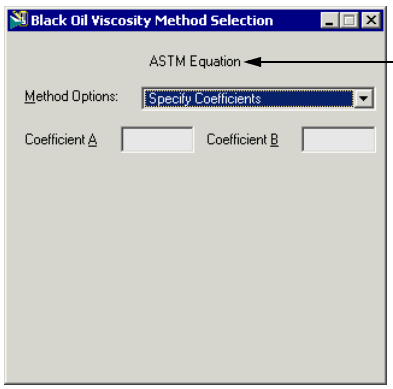
$$\mathrm{WOR} = \frac{V_{water}}{V_{oil}} \tag{1.2}$$

You can select your water content input preference from the drop-down list.

Next you will specify a method for calculating the dead oil viscosity.

13. Click on the **Viscosity Mtd** button. The Black Oil Viscosity Method Selection view appears.

Figure 1.19



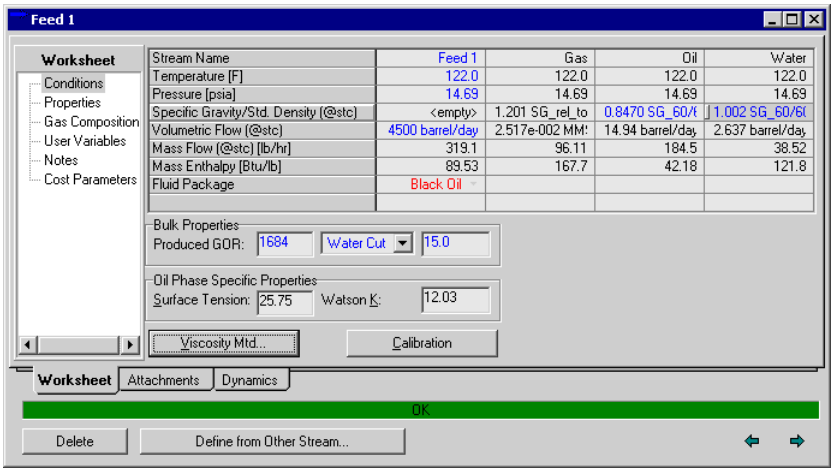
Displays the current selection of the Dead Oil Viscosity Equation. You can change this equation in the Neotec Black Oil Methods Manager. Refer to Dead Oil Viscosity Equation in **Appendix A.1 - Neotec Black Oil Methods and Thermodynamics** for more information.

You can select the calculation methods from the Method Options drop-down list. Neotec recommends the user to enter two or more viscosity data points. In the event that only one data point is known, this is also an improvement over relying on a generalized viscosity prediction.

14. Click on the **Method Options** drop-down list and select **Twu**.
15. Close the Black Oil Viscosity Method Selection view.

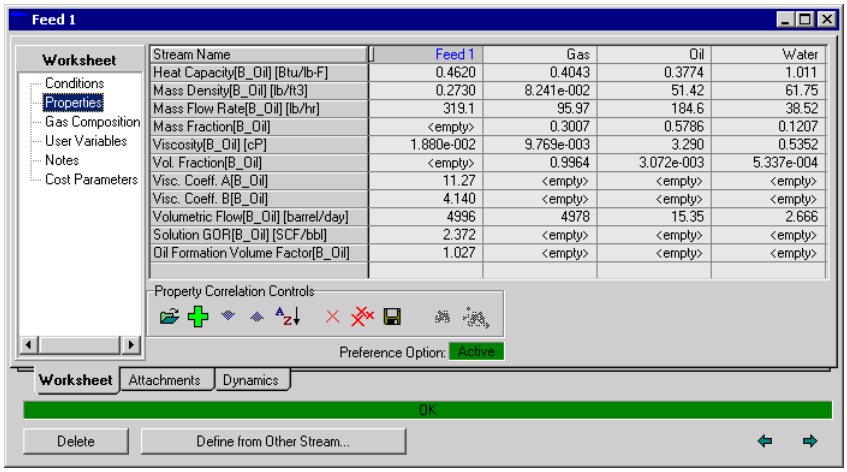
Now **Feed 1** is fully defined.

Figure 1.20



The Surface Tension and Watson K are automatically calculated by HYSYS as specified in the Neotec Black Oil Methods Manager. You can view the property correlations for each phase by clicking on the **Properties** page where you can add and delete correlations as desired.

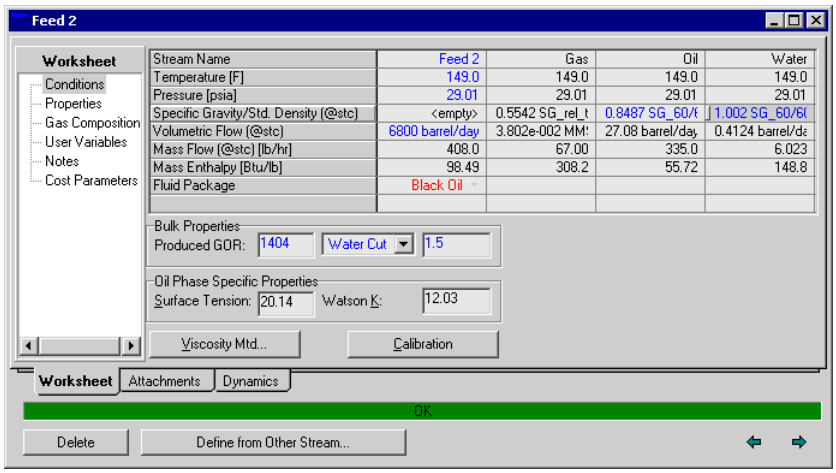
Figure 1.21



16. Create a second black oil feed stream, **Feed 2** and define it with the following data:

In these cells...	Enter...
Conditions Page	
Temperature (°F), Overall	149
Pressure (psia), Overall	29.01
Volumetric Flow (barrel/day), Overall	6800
Specific Gravity (SG_60/60 api)	Oil: 0.8487 Water: 1.002
Gas Oil Ratio	1404 SCF/bbl
Water Cut	1.5
Viscosity Method Options	Beggs and Robinson
Gas Composition Page	
Methane	1.0

Figure 1.22



1.4.2 Installing Unit Operations

The following unit operations can support black oil streams:

- Valve
- Mixer
- Pump
- Recycle
- Separator
- Pipe Segment
- Heat Exchanger
- Expander
- Compressor
- Heater
- Cooler

HYSYS unit operations typically solve to equilibrium conditions using an equation of state or activity model. With the HYSYS Upstream Option, HYSYS unit operations will solve in black oil mode and be able to blend different black oils together.

The Worksheet tab of some HYSYS unit operation property views are not supported when the unit operations are used in Black Oil mode.

Now you have fully defined two black oil feed streams. The next step is to install the necessary unit operations for the blending and transitioning process.

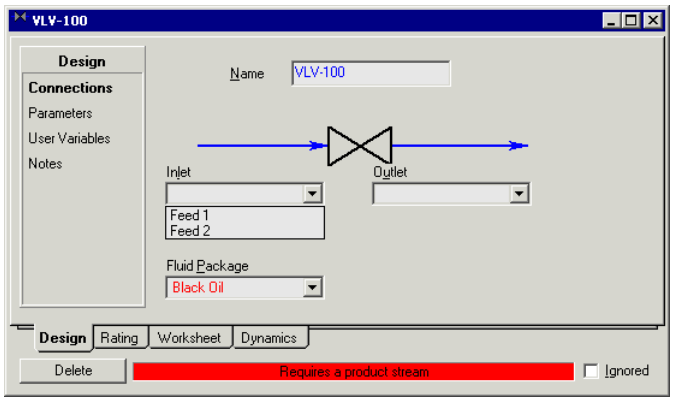
Installing the Valve

Valve icon

The first operation that will be installed is a Valve, used to decrease the pressure of Feed 1 before it is blended with Feed 2.

1. Double-click on the **Valve** icon in the Object Palette. The Valve property view appears.
2. On the **Connections** page, open the **Inlet** drop-down list by clicking the down arrow icon .

Figure 1.23



3. Select **Feed 1** from the list.

   Alternatively, you can make the connections by typing the exact stream name in the cell, then pressing **ENTER**.

4. Move to the **Outlet** field by clicking on it. Type **ValveOut** in the **Outlet** cell and press **ENTER**.

   The status indicator displays **Unknown Delta P**. To specify a pressure drop for the Valve:

5. Click on the **Parameters** page.
6. Specify **5 kPa** in the **Delta P** field.

Now the status indicator has changed to green **OK**, showing that the valve operation and attached streams are completely calculated.

Installing the Mixer

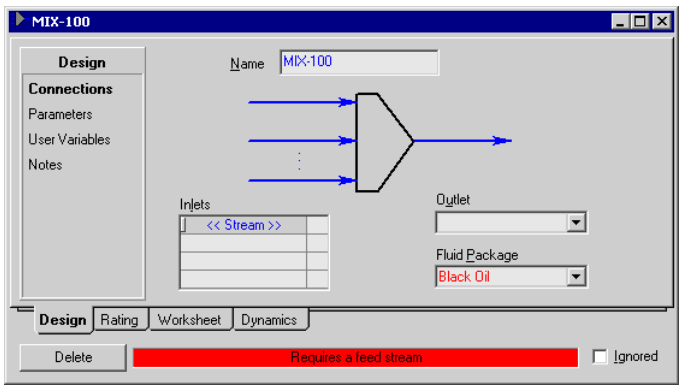
The second operation that will be installed is a Mixer, used to blend the two black oil feed streams.

To install the Mixer:

Mixer icon

1. Double-click on the **Mixer** icon in the Object Palette. The Mixer property view appears.

Figure 1.24



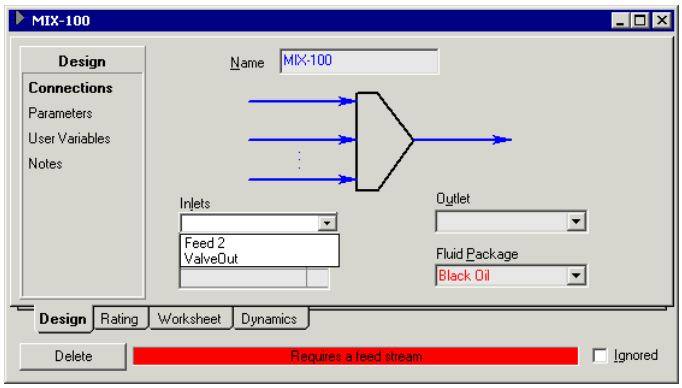
2. Click the **<<Stream>>** cell to ensure the **Inlets** table is active.

The status bar at the bottom of the view shows that the operation requires a feed stream.


Down arrow icon

3. Open the **<<Stream>>** drop-down list of feeds by clicking the down arrow icon, or by pressing **F2** and then the **DOWN** arrow key.

Figure 1.25



4. Select **ValveOut** from the list. The stream is transferred to the list of Inlets, and <<**Stream**>> is automatically moved down to a new empty cell.

Alternatively, you can make the connections by typing the exact stream name in the cell, then pressing **ENTER**.

5. Repeat steps 3-4 to connect the other stream, **Feed 2**.

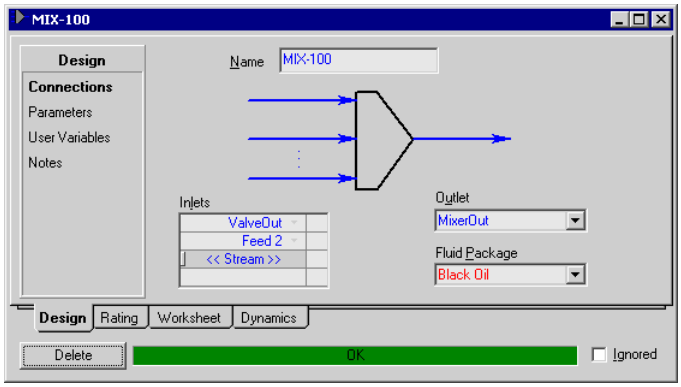
   The status indicator now displays **Requires a product stream**. Next you will assign a product stream.

6. Move to the **Outlet** field by clicking on it, or by pressing **TAB**.
7. Type **MixerOut** in the cell, then press **ENTER**.

   HYSYS recognizes that there is no existing stream named MixerOut, so it will create the new stream with this name.

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

Figure 1.26

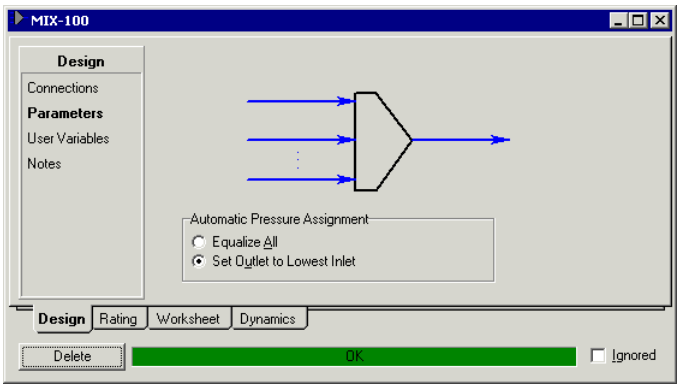


8. Click the **Parameters** page.
9. In the Automatic Pressure Assignment group, leave the default setting at **Set Outlet to Lowest Inlet**.

HYSYS has calculated the outlet stream by combining the two inlets and flashing the mixture at the lowest pressure of the inlet streams. In this case, ValveOut has a pressure of 96.3 kPa and Feed 2 has a pressure of 200 kPa. Thus, the outlet from the Mixer has a pressure of 96.3 kPa (the lowest pressure between the two inlets).

Refer to **Appendix A - Neotec Black Oil Methods**, for more information on the specific gravity and viscosity of heavy oil/ condensate blends.

Figure 1.27



Installing the Black Oil Translator

Next you will install a Black Oil Translator to transfer the black oil stream data into a compositional stream so that you can analyze the properties of the blended black oil stream from the Mixer. The Black Oil Translator is implemented in HYSYS using the Stream Cutter operation and a custom Black Oil Transition. The Black Oil Translator interacts with an existing Stream Cutter unit operation to convert the Black Oil stream into a compositional material stream.

Adding Non-Black Oil Stream

Before you install the Black Oil Translator, you need to install a non-black oil stream for the Black Oil Translator outlet stream. Thus, you will need to add a new fluid package and assign it to the outlet stream.

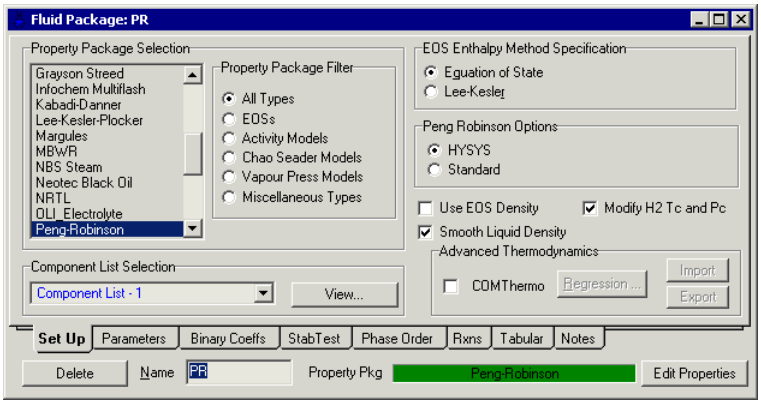
To add a new fluid package:

Enter Basis Environment icon

1. Click on the **Enter Basis Environment** icon in the tool bar. The Simulation Basis Manager appears.
2. Click on the **Fluid Pkgs** tab.
3. Click **Add**.
4. Select **Peng-Robinson** from the property package list in the Property Package Selection group.

5. In the **Name** field, rename the fluid package to **PR** as shown below.

Figure 1.28



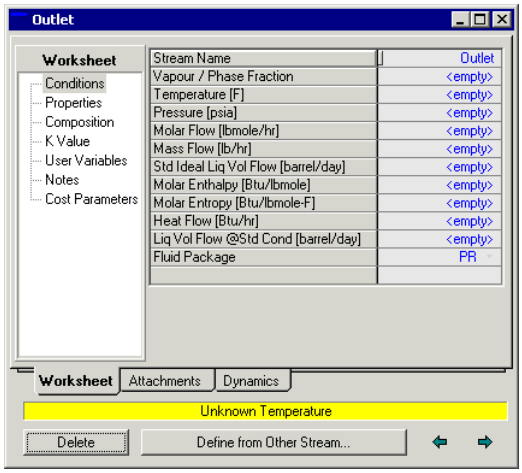
6. Close the Fluid Package view.
7. Click on the **Return to Simulation Environment** button in Simulation Basis Manger.
8. To add the Black Oil Translator outlet stream, do one of the following:
   - From the **Flowsheet** menu, select **Add Stream**.
   - Press **F11**.
   - From the Object Palette, double-click on the **Material Stream** icon.

Material Stream icon

9. In the stream property view, click the **Worksheet** tab and select the **Conditions** page.
10. In the **Stream Name** cell type **Outlet**.
11. In the **Fluid Package** cell, select **PR** from the drop-down list.

Once you selected PR as the fluid package, the Outlet stream property view is automatically changed to a HYSYS compositional stream.

Figure 1.29



12. Close the Outlet property view.

Adding the Black Oil Translator

There are two ways that you can add the Black Oil Translator to your simulation:

You can also open the UnitOps view by pressing the **F12** hot key.

1. From the **Flowsheet** menu, select **Add Operation**. The UnitOps view appears.
2. In the Categories group, select the **All Unit Ops** radio button.
3. From the Available Unit Operation lists, select **Black Oil Translator**.
4. Click **Add**.

OR

Upstream Ops icon

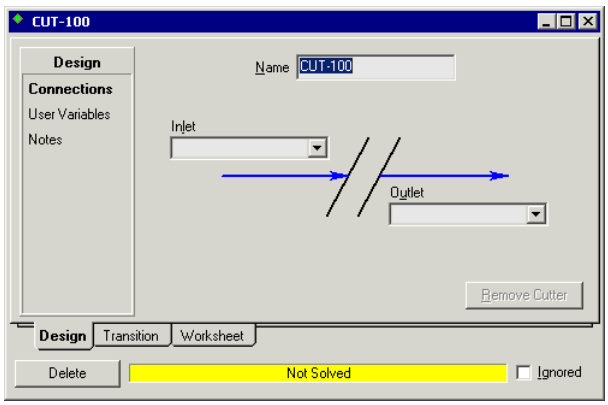
1. From the Object Palette, click on the **Upstream Ops** icon. The Upstream Ops Palette appears.

Figure 1.30

Black Oil Translator icon

2. Double-click the **Black Oil Translator** icon. The Black Oil Translator property view appears.

Figure 1.31



In certain situations, the Black Oil Translator will automatically be added to the flowsheet. This occurs when the stream connections are made to operations that have streams with different fluid packages connected or the operation itself is set to use a different fluid package. The Stream Cutter dictates the rules for when the Black Oil Translator is automatically added.

You can also delete a Black Oil Translator by clicking on the **Black Oil Translator** icon on the PFD and pressing the **DELETE** key.

To delete the Black Oil Translator operation, click the **Delete** button. HYSYS will ask you to confirm the deletion.

To ignore the Black Oil Translator operation during calculations,

activate the Ignored checkbox. HYSYS completely disregards the operation (not calculate the outlet stream) until you restore it to an active state by deactivating the checkbox.

Defining the Black Oil Translator

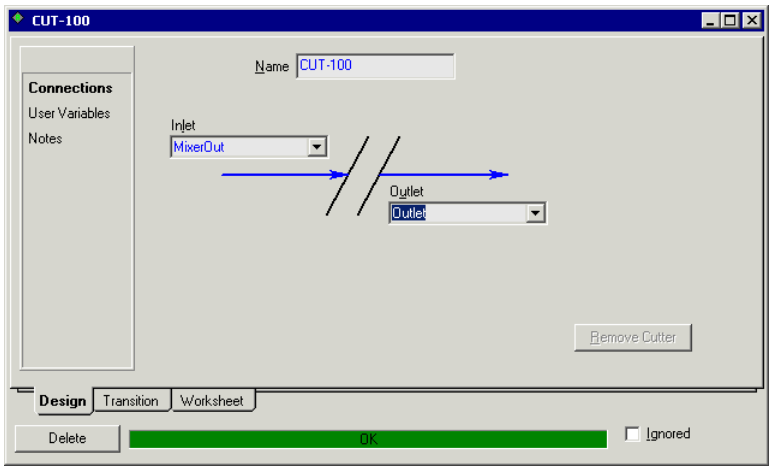
To complete the Connections page:


Down arrow icon

1. Open the **Inlet** drop-down list by clicking the down arrow icon, or by pressing the **F2** key and then the **DOWN** arrow key.
2. Select **MixerOut** as the inlet.
3. Move to the **Outlet** field by clicking on it.
4. Select **Outlet** as the outlet stream.

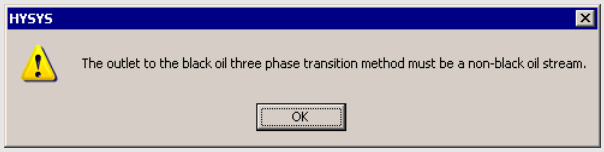
   Once the outlet stream is connected, the Black Oil Translator starts transitioning the black oil data to the **Outlet** stream using the HYSYS default transition setting.

Figure 1.32



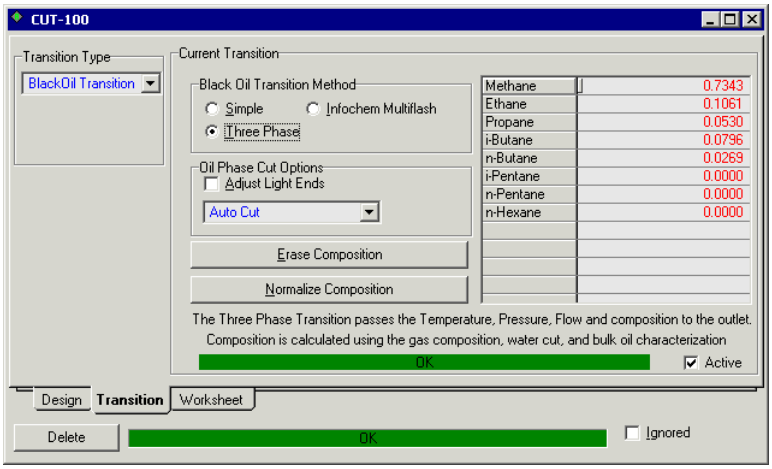
The solving status is indicated in the Object Status Window. As the Black Oil Translator is solving, a list of hypocomponents are generated in the Outlet stream to characterize a black oil stream from a compositional stream perspective. You can view each hypocomponent created in the Trace Window as the Black Oil Translator is solving.

If the Outlet stream had Black Oil as the fluid package, the following warning message view would appear.



5. Click on the **Transition** tab.

Figure 1.33



- The Transition Type group displays the transition type (in this case: **BlackOil Transition**) available for this Black Oil Translator operation.
- The Current Transition group contains all the options used to configure the Black Oil Transition method.

The composition of **MixerOut** is copied to the composition table as shown in the figure above. Leave the composition as default.

Refer to **Appendix B - Black Oil Transition Methods** for more information on the Simple, Three Phase, and Infochem Multiflash transition method.

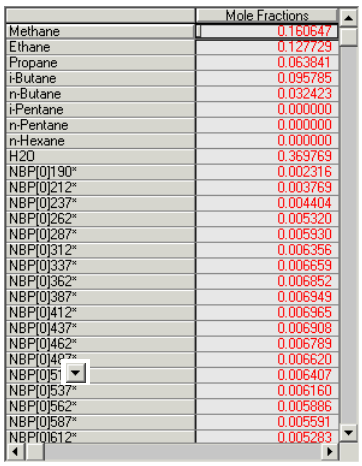
6. In the Black Oil Transition Method group, confirm that the **Three Phase** radio button is selected.
7. Save the case.

1.4.3 Results

When the solving is completed, the status indicator for the Outlet stream and Black Oil Translator should be changed to a green OK, showing that both operations are completely defined.

1. In the Outlet stream property view, click on the **Compositions** page on the **Worksheet** tab.
2. In the component composition list, you can view the composition for all the hypocomponents created as well as the composition for C1 to C6.

Figure 1.34



3. Close the Outlet stream property view.
4. Double-click on the **CUT-100** operation on the PFD. The black oil translator property view appears.
5. Click on the **Worksheet** tab.

   On the **Conditions** page, the Compositional stream properties and conditions for the black oil stream MixerOut are displayed in the Outlet column.

CUT-100 operation

You can examine and review the results for the MixerOut stream as a compositional stream.

Figure 1.35

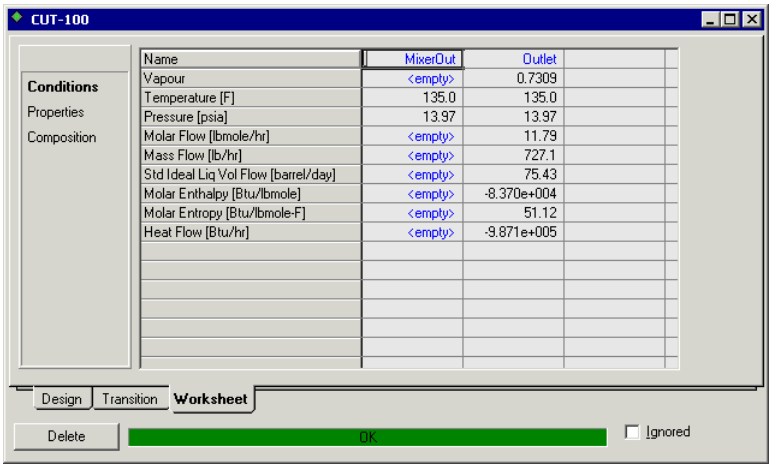
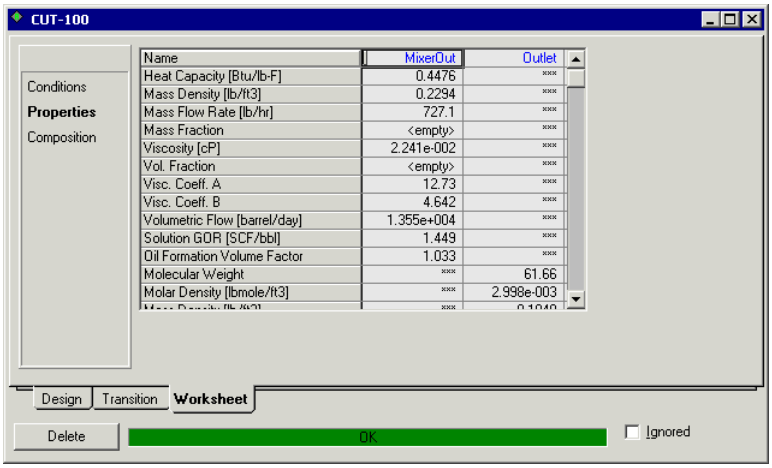


Figure 1.36



1.5 Dynamic Simulation

In this tutorial, the black oil dynamic capability will be incorporated into a steady-state black oil translation simulation case.

You can continue into this dynamic section with the black oil translation case that you built during the steady state section.

1. Open the **BlackOil.hsc** case (if it is not already open in the HYSYS).
2. Save the case under the new name: **BlackOilDyn.hsc**.

1.5.1 Modifying the Steady State Flowsheet

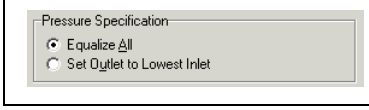
Before the case can be run in Dynamic mode, it is necessary to modify the steady-state model so that a pressure-flow relation exists between each unit operation.

In order to realistically model flow behaviour in a dynamic simulation case, you will change the Mixer to equalize all inlet pressures so that the flow to and from the Mixer is determined by the pressure-flow network:

1. Delete the specified pressure in **Feed 2**.
2. Double-click on the **Mixer**. The Mixer property view appears.
3. Click on the **Dynamics** tab, and select the **Specs** page.
4. In the Pressure Specification group, select the **Equalize All** radio button.

You can also equalize the inlets pressures for the Mixer by selecting the **Equalize All** radio button on the **Parameters** page on the **Design** tab.

Figure 1.37



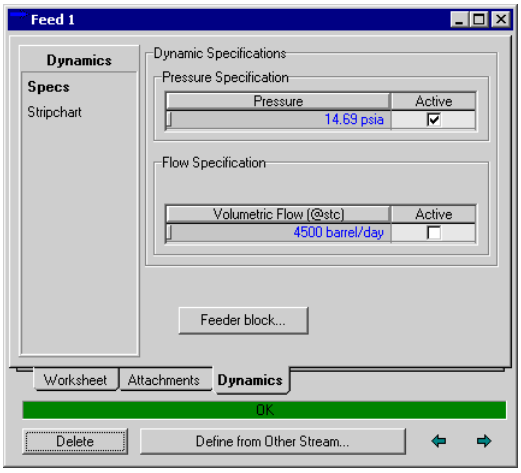
HYSYS automatically recalculates and solves the Mixer operation. The status indicator of the Mixer has now changed to a green **OK**. The flowsheet is completely defined.

1.5.2 Setting Pressure-Flow Specifications

In Dynamic mode, the degrees of freedom for the flowsheet must be zero. You can reduce the degrees of freedom by setting the pressure-flow specifications in each boundary stream:

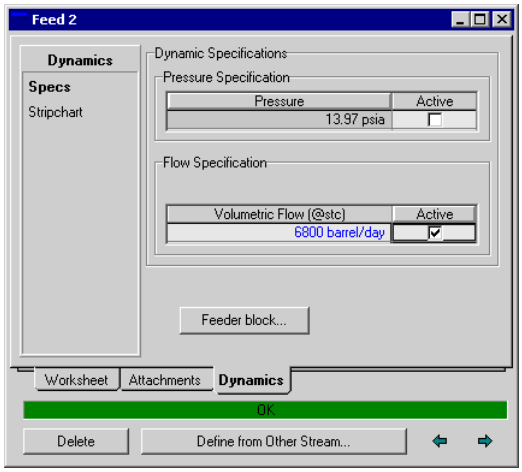
1. Double-click on **Feed 1**. The stream property view appears.
2. Click on the **Dynamics** tab, and select the **Specs** page.
3. Ensure the Pressure specification is active with a check in Active checkbox, and deactivate the Volumetric Flow specification.

Figure 1.38



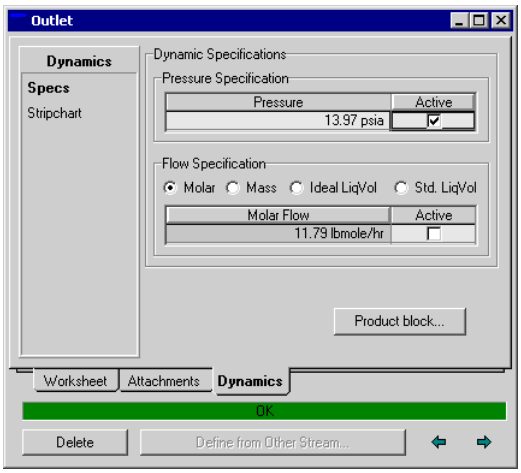
4. For **Feed 2**, specify the **Specs** page as shown:

Figure 1.39



5. For **Outlet**, specify the pressure-flow specifications as shown:

Figure 1.40



6. Save the case. The simulation case is ready to run in Dynamic mode.


Integrator Holding icon (red)


Dynamic Mode icon

7. Click the **Integrator Holding** icon (red) on the Tools bar to hold all calculations.
8. Click the **Dynamics Mode** icon on the Tools bar.
9. A view appears asking you to confirm switching the simulation case to Dynamics mode. Click **Yes**.

   If the Dynamics Assistant is active, HYSYS will ask you whether you want to make certain changes to the simulation case in the Dynamic Assistant before engaging in dynamic mode. Click **No** to the Dynamic Assistant.

The Dynamic Assistant is one of the methods for preparing a steady state case for dynamic mode. You can set your own pressure-flow specifications and size the unit operations manually on their Specs page on the Dynamics tab.

   The Dynamics Assistant makes **recommendations** as to how the flowsheet topology should change and what pressure-flow specifications are required in order to run the case in dynamic mode. However, in this tutorial some of these changes have been made manually as you modified the flowsheet, and the remaining changes are not necessary for the purpose of this example.


Integrator Active icon (green)

10. Start the Integrator by clicking the **Integrator Active** icon in the toolbar.

The simulation case is now running in Dynamic mode. The integration time and status are indicated in the Trace Window and Status Bar.

1.5.3 Monitoring in Dynamics

In Dynamic mode it is difficult to observe the behaviour of simulation variables as they vary with time. Stripchart allows you to monitor various variable sets of interest as they are constantly updated in real time. You will create a strip chart to monitor the temperature, pressure, and flow for the Outlet:

Integrator Holding icon (red)

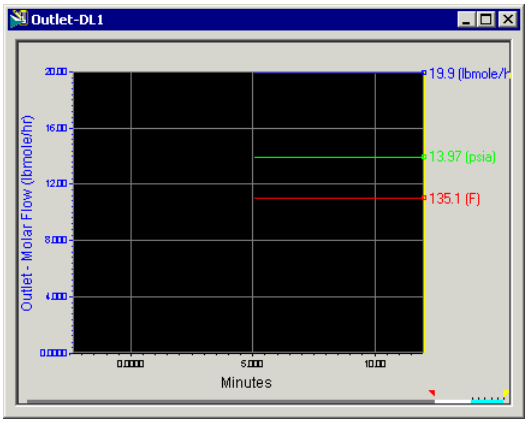
1. Set the Integrator to holding mode by clicking the **Integrator Holding** icon.
2. Double-click on the **Outlet** stream.
3. Click on the **Dynamics** tab, and select the **Stripchart** page.
4. From the **Variable Set** drop-down list, select the **T, P, and F** variables set.
5. Display the strip chart by clicking the **Create Stripchart** button. A strip chart view appears.

Integrator Active icon (green)

6. Activate the Integrator by clicking the **Integrator Active** icon in the toolbar.

   As the Integrator is running, you should see the temperature, pressure, and flow of the Outlet updating.

Figure 1.41



1.5.4 Notes

The following should be noted when using black oil in Dynamic mode:

- Black oil system does not support Component Splitter and tray section since they are strongly linked to composition.
- In steady-state black oil translation, the component list changes after the black oil stream is converted to a compositional stream. However for black oil translation in dynamic mode, a new composition using the existing component list is calculated. Ensure that the desired components are already present on the non-black oil side of the transition before the simulation starts. The simplest way to do this is to use the component list from a steady-state result.
- Always refer to the stream property view for the black oil simulation information.
- To obtain the most accurate black oil results, avoid using black oil system with extreme simulation conditions or phase ratios.

A Neotec Black Oil Methods

A.1 Neotec Black Oil Methods and Thermodynamics

You can select the desired black oil methods in the Neotec Black Oil Methods Manager.

Several black oil PVT calculation methods exist, each based on data from a relatively specific producing area of the world.

Correlations	Data
Standing (1947) Correlation for R_S and B_O	Based on 22 California crude oil-gas systems.
Lasater (1958) Correlation for R_S	Developed using 158 data from 137 crude-oils from Canada, Western and mid-continent USA, and South America.
Vasquez and Beggs (1977) Correlations for R_S and B_O	Based on 6004 data. Developed using data from Mid-West and California crudes.
Glaso (1980) Correlations for R_S and B_O	For volatile and non-volatile oils. Developed using data from North Sea crudes.
Al-Marhoun (1985, 1988, 1992) Correlations for R_S and B_O	Based on data from Saudi crude oils and Middle East reservoirs.
Abdul-Majeed and Salman (1988) Correlation for B_O	Based on 420 data points from 119 crude oil-gas systems, primarily from Middle East reservoirs.
Dokla and Osman (1992) Correlations for R_S and B_O	Based on 51 bottomhole samples taken from UAE reservoirs.
Petrosky and Farshad (1993) Correlations for R_S and B_O	Based on 81 oil samples from reservoirs in the Gulf of Mexico.

A.1.1 Terminology

Before we discuss the PVT behaviour and transport property procedures, you should be familiar with the following terms:

- Stock Tank Conditions
- Produced Gas Oil Ratio
- Solution Gas Oil Ratio
- Viscosity of Heavy Oil/Condensate Blends
- Specific Enthalpies for Gases and Liquids
- Oil-Water Emulsions

Stock Tank Conditions

Stock tank conditions are the basic reference conditions at which the properties of different hydrocarbon systems can be compared on a consistent basis. The stock tank conditions are defined as 14.70 psia (101.325 kPa) and 60°F (15°C).

Produced Gas Oil Ratio

The produced gas oil ratio is the total amount of gas that is produced from the reservoir with one stock tank volume of oil. Typical units are scf/stb or m^3 at s.c./m^3 at s.c.

Solution Gas Oil Ratio

The solution gas/oil ratio is the amount of gas that saturates in the oil at a given pressure and temperature. Typical units are scf/stb or m3at s.c./m3 at s.c.

Above the bubble point pressure, for a given temperature, the solution gas/oil ratio is equal to the produced gas oil ratio. For stock tank oil (i.e., oil at stock tank conditions) the solution gas oil ratio is considered to be zero.

Viscosity of Heavy Oil/Condensate Blends

A common relationship for estimating the viscosity of a mixture of two hydrocarbon liquids is as follows:

$$\mu_m = \mu_A^{C_A} \times \mu_B^{(1-C_A)} \quad \text{(A.1)}$$

where:

μ_m= *viscosity of the blended stream*

μ_A = *viscosity of liquid A*

μ_B = *viscosity of liquid B*

C_A= *volume fraction of liquid A in the blended stream*

For cases where $\frac{\mu_A}{\mu_B} > 20$, it is recommended by Shu (1984) that another correlation should be used to calculate the viscosity of the mixture assuming that liquid A is the heavier and more viscous fluid than liquid B.

$$\mu_m = \mu_A^{X_A} \times \mu_B^{(1-X_A)} \quad \text{(A.2)}$$

where:

$$X_A = \frac{\alpha C_A}{\alpha C_A + C_B} \quad \text{(A.3)}$$

$$\alpha = \frac{17.04(S_A - S_B)^{0.5237} S_A^{3.2745} S_B^{1.6316}}{\mathrm{Ln}\left(\frac{\mu_A}{\mu_B}\right)} \quad \text{(A.4)}$$

S_A = *specific gravity of liquid A*

S_B = *specific gravity of liquid B*

Data from two different crude oil/condensate blends have been used to compare the results predicted by **Equation (A.1)** and **Equation (A.2)** through **Equation (A.4)**. The following table contains the available data for the two oils and the condensate liquid.

Liquid	API Gravity	Specific Gravity	Viscosity (mPa.s)		
			5°C	10°C	20°C
Oil A	14.3	0.970	12840	7400	2736
Oil B	14.3	0.964	3725	2350	1000
Condensate	82.1	0.662	0.42	0.385	-

To simplify viscosity calculations at intermediate temperatures, the data given in the above table for each liquid were fitted to the following form:

$$\mu = a \cdot \left(\frac{100}{1.8 \cdot T + 32} \right)^{b} \qquad \text{(A.5)}$$

where:

T = temperature, °C

a, b = fitted constants

The resulting values of a and b are given in the following table.

Liquid	a	b
Oil A	849.0	3.07
Oil B	370.0	2.62
Condensate	0.28	0.44

In all cases, the fit is very accurate (maximum error is about 3.6%) and the use of **Equation (A.5)** introduces minimal error into the comparison.

Measured data were available at three temperatures (0°C, 5°C, and 10°C) for each of the crude oils with three blending ratios (90%, 80%, and 70% crude oil). Mixture viscosities calculated by **Equation (A.1)** and **Equation (A.2)** are compared with these data in the following table.

Oil	Temp (°C)	Blend (% crude)	μ_{meas} (mPa.s)	Equation 1.3		Equation 1.4	
				μ_{calc} (mPa.s)	error (%)	μ_{calc} (mPa.s)	error (%)
A	0	90	2220	9348	321.1	2392	7.8
A	0	80	382	3111	714.4	370	-3.1
A	0	70	89	1035	1062.9	86	-3.4
A	5	90	1464	4661	218.4	1442	-1.5
A	5	80	272	1656	508.2	260	-4.4
A	5	70	71	588	728.2	66	-7.0
A	10	90	976	2670	173.6	953	-2.4
A	10	80	198	999	404.6	194	-2.0
A	10	70	56	374	567.9	53	-5.4
B	0	90	744	2774	272.9	989	32.9
B	0	80	147	1056	618.4	205	39.5
B	0	70	45	402	793.3	58	28.9
B	5	90	516	1531	196.7	629	21.9
B	5	80	112	615	449.1	148	32.1
B	5	70	37	247	567.6	45	21.6
B	10	90	396	951	140.2	436	10.1
B	10	80	87	399	358.6	113	29.9
B	10	70	29	168	479.3	37	27.6

From the table it is clear that the results calculated using **Equation (A.1)** are not acceptable and would lead to gross errors calculated pressure losses. As for **Equation (A.2)**, it gives excellent results for the blends involving Oil A. While the errors associated with Oil B blends are significantly larger, they are not unreasonable.

Equation (A.3) can be further modified to improve its accuracy by introducing a proprietary calibration factor.

Oil	Temp (°C)	Blend (% crude)	μ_{meas} (mPa.s)	Equation 1.4	
				μ_{calc} (mPa.s)	error (%)
B	0	90	744	817	9.8
B	0	80	147	157	6.8
B	0	70	45	43	-4.4
B	5	90	516	529	2.5
B	5	80	112	115	2.7
B	5	70	37	34	-8.1
B	10	90	396	371	-6.3
B	10	80	87	90	3.4
B	10	70	29	28	-3.4

The results obtained from the modified Shu correlation show that the calibration procedure has yielded a significant improvement in accuracy. This also applies to data at 0°C, which were not used in the determination of the calibration since no measured viscosity values for either Oil B or the condensate were available at that temperature.

It has been demonstrated that the correlation of Shu (1984) is much superior to the simple blending relationship expressed by **Equation (A.1)**, and it is capable of giving acceptable accuracy for most pipeline pressure drop calculations.

Specific Enthalpies for Gases and Liquids

In pipelines and wells the Joule-Thompson effect is typically exhibited as a large decrease in temperature as a gas expands across a restriction. According to the relationships between the temperature, pressure, and latent energy of the fluid, the fluid typically cools when it expands, and warms when compressed.

The temperature profiles are calculated by simultaneously solving the mechanical and total energy balance equations. The latter includes a term that is directly related to changes in the total enthalpy of the fluid(s). This means that all Joule-Thompson expansion cooling effects for gases, and frictional heating effects for liquids would be taken into account implicitly. It is not necessary, for example, to impose the approximations inherent in specifying a constant average value of a Joule-Thompson coefficient. it is, however, necessary to be able to compute the specific enthalpy of any gas or liquid phase, at any

pressure and temperature, as accurately as possible. The following sections describe the procedures for computing this important thermodynamic parameter for various fluid systems.

Undefined Gases

For undefined single phase gases, where only the gravity is known, the specific enthalpy is determined by assuming the gas to be a binary mixture of the first two normal hydrocarbon gases whose gravities span that of the unknown gas. The mole fractions are selected such that the gravity of the binary mixture is identical to that of the unknown gas of interest.

For example, a natural gas having a gravity of 0.688 would be characterized as a binary mixture consisting of 72.3 mole % methane (gravity = 0.5539) and 27.7 mole % ethane (gravity = 1.0382) since (0.723)(0.5539) + (0.277)(1.0382) = 0.688. The enthalpy of the binary mixture, calculated as described above for compositional systems, is then taken as the enthalpy of the gas of interest. This is in fact the same procedure that has been used to create the generalized specific enthalpy charts that appear in the GPSA Engineering Data Book (1987).

The specific enthalpy has been evaluated as described above for a number of specified gas gravities over a relatively wide range of pressures and temperatures. The enthalpy of the unknown gas is obtained at any given pressure and temperature by interpolation within the resulting matrix of values.

Undefined Liquids

Undefined hydrocarbon liquids are characterized only by a specific or API gravity, and possibly also the Watson K factor. They are also referred to as "black oils", and the specific enthalpy is computed using the specific heat capacity calculated using the correlation of Watson and Nelson (1933):

$$C_p = A_1 \times [A_2 + (A_3 T)] \tag{A.6}$$

where:

C_p = *specific heat capacity of the oil, btu/lb°F*

T = *temperature, °F*

The three coefficients have the following equations:

$$\begin{aligned} A_1 &= 0.055K + 0.35 \\ A_2 &= 0.6811 - 0.308\gamma_o \\ A_3 &= 0.000815 - 0.000306\gamma_o \end{aligned} \tag{A.7}$$

where:

K = *Watson K factor* = $\dfrac{T_B^{1/3}}{S_o}$

S_o = *specific gravity of the oil*

The specific enthalpy at any temperature T, relative to some reference temperature T_o, is given by the following equation:

$$H = \int_{T_o}^{T} C_p(T)\,dT \tag{A.8}$$

The specific enthalpy computed using **Equation (A.8)** is independent of pressure. For real liquids, the effect of pressure is relatively small compared to the temperature effect, but it may become significant when the pressure gradient is large due to flow rate rather than elevation effects.

Large pressure gradients tend to occur with high viscosity oils. At higher flow rates, frictional heating effects can become significant, and the heating tends to reduce the oil viscosity, which in turn, affects the pressure gradient. Unfortunately, this complex interaction cannot be predicted mathematically using specific enthalpy values that are independent of pressure. The net result is that the predicted pressure gradient will be higher than should actually be expected.

For fully compositional systems, the calculated specific enthalpy of a liquid phase does include the effect of pressure. A series of calculations have been performed using the Peng-Robinson (1976) equation of state for a variety of hydrocarbon liquids, ranging from relatively light condensate liquids to relatively heavy crude oils. In each case, specific enthalpy was calculated over a wide range of pressures at a low, moderate, and high temperature. In the case of the condensate liquids, specific compositional analyses were used. For the heavier crude oils, the composition consisted of a number of pseudo-components, based on published boiling point assay data, as generated by Neotec's technical utility module HYPOS. In all cases, the effect of pressure was found to be constant and is well represented by the following relation:

$$H_{P,T} = H_{P^o,T} + 0.0038 \times (P - 15) \qquad \text{(A.9)}$$

where:

$H_{P,T}$ = *specific enthalpy at the specific pressure and temperature, btu/lb-°F*

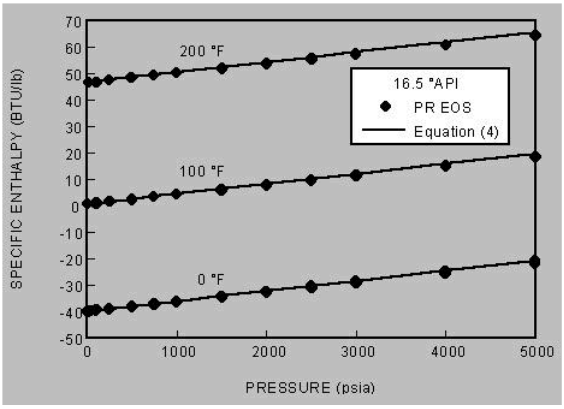
$H_{P^o,T}$ = *specific enthalpy computed with* ***Equation (A.8)***

P = *pressure, psia*

Figure A.1, **Figure A.2**, and **Figure A.3** show the comparison between specific enthalpies calculated using the Peng Robinson equation of state and those computed using **Equation (A.9)** for 16.5, 31.9, and 40.5° API oils, respectively. For comparison purposes, $H_{P^o,T}$ was taken to be the value computed by the Peng Robinson equations of state at 15 psia.

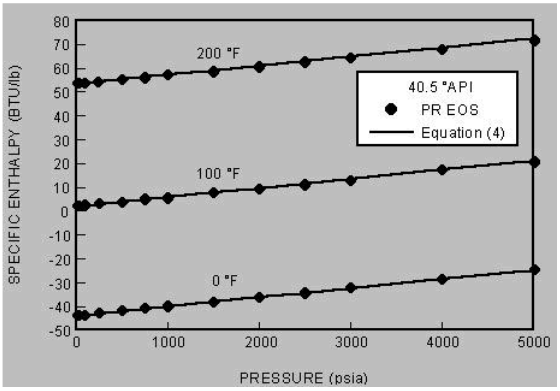
Effect of Pressure on Specific Enthalpy for a 16.5° API Oil

Figure A.1



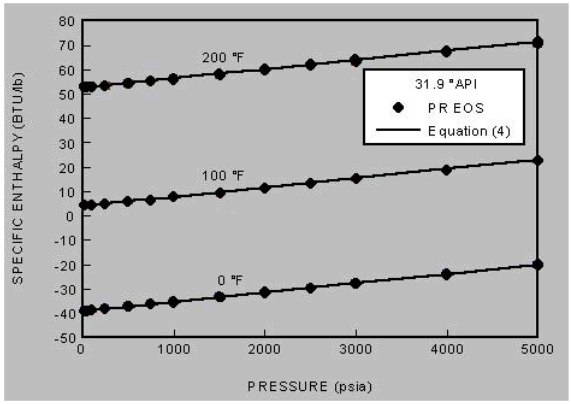
Effect of Pressure on Specific Enthalpy for a 40.5° API Oil

Figure A.2



Effect of Pressure on Specific Enthalpy for a 31.9° API Oil

Figure A.3



The effect of pressure is included in all specific enthalpy calculations, and therefore, in all temperature profile calculations, in a way that closely approximates similar calculations for fully compositional systems.

Oil-Water Emulsions

The rheological behaviour of emulsions may be non-Newtonian and is often very complex. Generalized methods for predicting transport properties are limited because of the wide variation in observed properties for apparently similar fluids. It is usually the case with non-Newtonian fluids that some laboratory data or other experimental observations are required to provide a basis for selecting or tuning transport property prediction methods.

Neotec assumed that an emulsion behaves as a pseudo-homogeneous mixture of hydrocarbon liquid and water and may thus be treated as if it were a single liquid phase with appropriately defined transport properties.

The volumetric flow rate of this assumed phase is the sum of the oil and water volumetric flow rates,

$$Q_e = Q_o + Q_w \tag{A.10}$$

where:

Q_e = *volumetric flow rate of emulsion, ft^3/sec or m^3/sec*

Q_o = *volumetric flow rate of oil, ft^3/sec or m^3/sec*

Q_w = *volumetric flow rate of water, ft^3/sec or m^3/sec*

The water volume fraction in the emulsion, C_w, is thus given by,

$$C_w = \frac{Q_w}{Q_o + Q_w} \tag{A.11}$$

Since the emulsion is assumed to be a pseudo-homogeneous mixture, the density is given by,

$$\rho_e = \rho_w C_w + \rho_o (1 - C_w) \tag{A.12}$$

where:

ρ_e = *density of the emulsion, lb/ft^3 or kg/m^3*

ρ_w = *density of the water at flowing conditions, lb/ft^3 or kg/ m^3*

ρ_o = *density of the oil at flowing conditions, lb/ft^3 or kg/m^3*

The effective viscosity of an emulsion depends on the properties of the oil, the properties of the water, and the relative amounts of each phase. For a water-in-oil emulsion (i.e. oil is the continuous phase), the effective viscosity of the emulsion can be much higher than that of the pure oil.

A commonly used relationship for estimating the viscosity of a water-in-oil emulsion is,

$$\mu_e = F_e \mu_o \tag{A.13}$$

where:

μ_e = *viscosity of the emulsion, cP or mPa.s*

μ_o = *viscosity of the oil, cP or mPa.s*

F_e = *emulsion viscosity factor*

The factor Fe is usually considered to be a function of the water fraction C_w and the best known procedure for estimating F_e is the graphical correlation of Woelflin (1942).

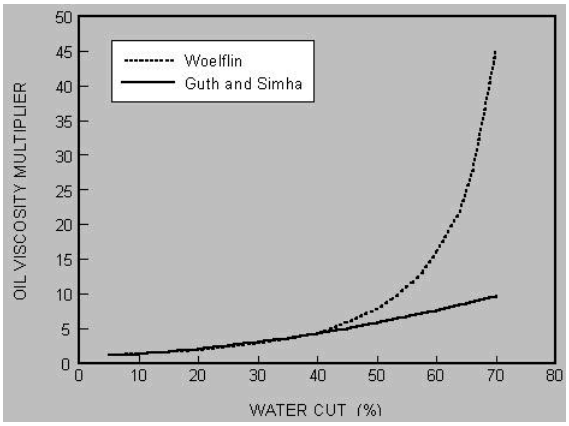
More recently, Smith and Arnold (see Bradley, 1987) recommended the use of the following simple quadratic equation,

$$F_e = 1.0 + 2.5 C_w + 14.1 C_w^2 \tag{A.14}$$

The emulsion viscosity factors based on Woelflin's 'medium' emulsion curve (he also presented curves for 'loose' and 'tight' emulsions) are compared in **Figure A.4** with those calculated using **Equation (A.14)**.

The two relationships are virtually identical for $C_w < 0.4$, but diverge rapidly at higher values of C_w.

Figure A.4



With increasing water fraction, the system will gradually behave more like water than oil. The water fraction at which the system changes from a water-in-oil emulsion to an oil-in-water emulsion is called the inversion point. The transition to an oil-in-water emulsion is generally very abrupt and characterized by a marked decrease in the effective viscosity. The actual inversion point must usually be determined experimentally for a given system as there is no reliable way to predict it. In many cases however, it is observed to occur in mixtures consisting of between 50% and 70% water.

Guth and Simha (1936) proposed a similar correlation as Smith and Arnold (**Equation (A.14)**),

$$F_e = 1.0 + 2.5C_d + 14.1C_d^2 \qquad \text{(A.15)}$$

where:

F_e = *emulsion viscosity multiplier for the continuous phase viscosity*

C_d = *volume fraction of the dispersed phase*

If C_{wi} is defined as the water fraction at the inversion point, then for $C_w < C_{wi}$, the emulsion viscosity is given by **Equation (A.13)**, with F_e defined by **Equation (A.14)**. However, for $C_w > C_{wi}$, the emulsion viscosity should be computed using the following expression,

$$\mu_e = F_e \mu_w \tag{A.16}$$

where:

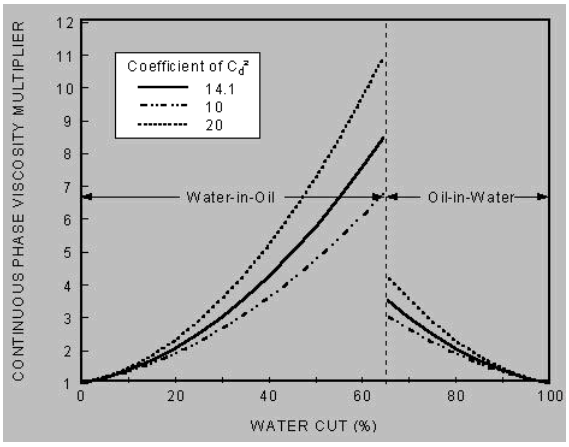
μ_w = *viscosity of the water phase, cP or mPa.s*

$F_e = 1.0 + 2.5(1-C_w)+14.1(1-C_w)^2$

As shown in **Equation (A.15)**, while the constant and the first order term on the right can be shown to have a theoretical basis, the squared term represents a purely empirical modification. It seems reasonable therefore to view the coefficient of the squared term (i.e., 14.1) as an adjustable parameter in cases where actual data are available.

To illustrate the predicted effect of the inversion point, **Figure A.5** shows a case in which C_{wi} = 0.65. Also the corresponding curves for several different values of the coefficient of the squared term are compared.

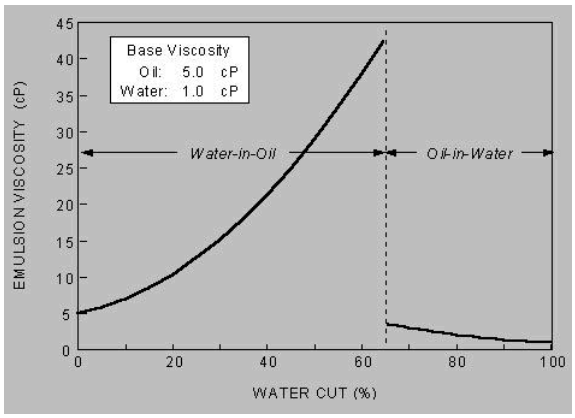
Figure A.5



The large decrease in the predicted value of the emulsion viscosity is evident. The effect on the emulsion viscosity can be seen in **Figure A.6**, since, above the inversion point, the factor is used to multiply the water viscosity, which is typically significantly lower than the oil viscosity.

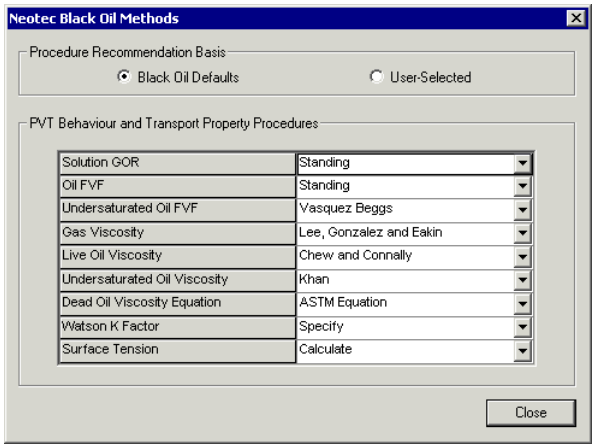
Limited experience to date in performing pressure loss calculations for emulsions suggests that the Woelflin correlation over-estimates the viscosity at higher water fractions. It is thus recommended that one use the Guth and Simha equation unless available data for a particular case suggest otherwise.

Figure A.6



A.1.2 PVT Behaviour and Transport Property Procedures

Figure A.7



There are nine PVT behaviour and transport property procedures available in the Neotec Black Oil Methods Manger:

- Solution GOR
- Oil FVF
- Undersaturated Oil FVF
- Gas Viscosity
- Live Oil Viscosity
- Undersaturated Oil Viscosity
- Dead Oil Viscosity Equation
- Watson K Factor
- Surface Tension

Solution GOR

The solution gas oil ratio, Rs, is the amount of gas that is assumed to be dissolved in the oil at a given pressure and temperature. Typical units are scf/stb or m^3 at s.c./m^3 at s.c.

Above the bubble point pressure, for a given temperature, the solution gas oil ratio is equal to the Produced Gas Oil Ratio. For the oil at Stock Tank Conditions, the solution gas oil ratio is considered to be zero.

You can select one of the following methods to calculate the solution GOR:

- Standing.
- Vasquez Beggs.
- Lasater.
- Glaso (Non Volatile Oils)
- Glaso (Volatile Oils)
- Al Marhoun (1985)
- Al Marhoun (Middle East Oils)
- Petrosky and Farshad
- Dolka and Osman

Oil FVF

The Oil Formation Volume Factor is the ratio of the liquid volume at stock tank conditions to that at reservoir conditions.

The formation volume factor (FVF, B_o) for a hydrocarbon liquid is the volume of one stock tank volume of that liquid plus its dissolved gas (if any), at a given pressure and temperature, relative to the volume of that liquid at stock tank conditions. Typical units are bbl/stb or m^3/m^3 at s.c.

You can select one of the following methods to calculate the Oil FVF:

- Standing
- Vasquez Beggs
- Glaso
- Al Marhoun (1985),
- Al Marhoun (Middle East OIls)
- Al Marhoun (1992)
- Abdul-Majeed and Salman
- Petrosky and Farshad
- Dolka and Osman

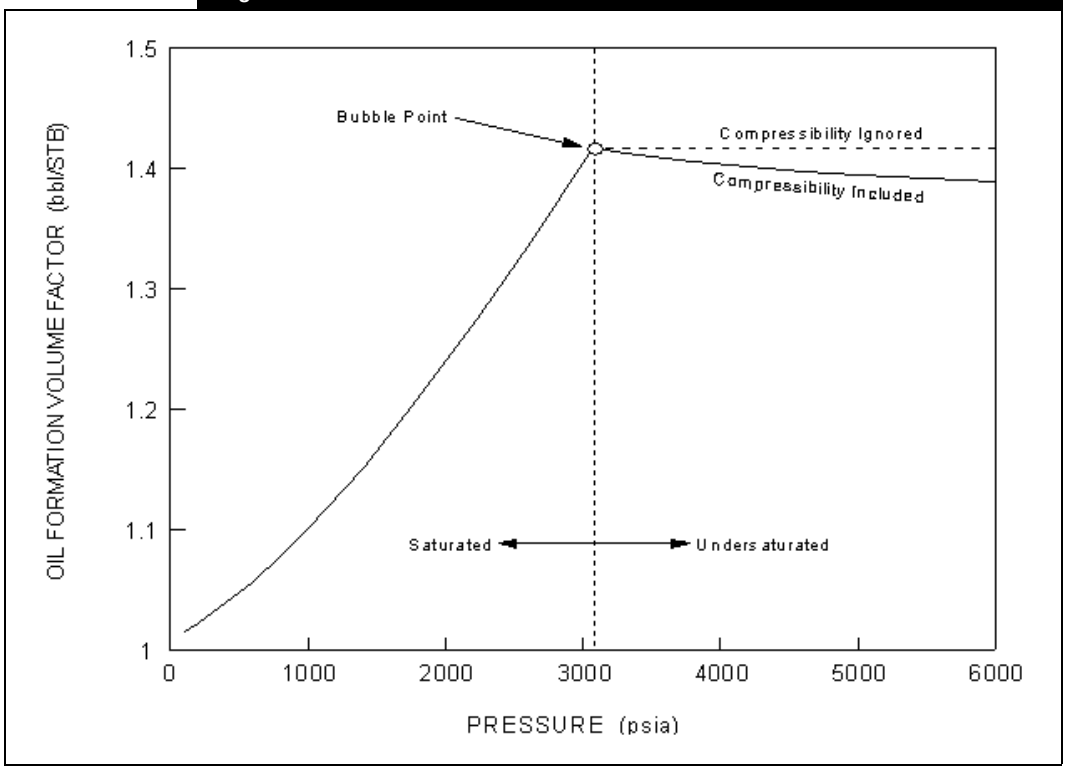
Undersaturated Oil FVF

In HYSYS, the default calculation method is Vasquez Beggs. You can choose other calculation methods as follows:

- Al Marhoun (1992)
- Petrosky and Farshad

Figure A.8 shows the typical behaviour of the oil formation volume factor that is observed as the system pressure is increased at a constant temperature.

Figure A.8



From the initial pressure up to the bubble point pressure (i.e., the point at which GOR = R_s, which happens to be 3,073 psia in this case), the oil is assumed to be saturated, and B_o continues to increase, as more and more gas goes into solution. The effect of this increasing solution gas is always much greater than the corresponding shrinkage of the oil due to pure compression effects.

At the bubble point, there is no more gas to go into the solution, and the oil then becomes progressively more undersaturated with increasing pressure. With the solution gas-oil ratio being constant, the portion of the curve in **Figure A.8** labelled "Compressibility Ignored" shows the behaviour that would be predicted by the correlations for B_o that we have looked at to this point. In actual fact, however, at pressures greater than the bubble point pressure, B_o is decreasing, due totally to the compressibility of the oil. The actual behaviour that is observed is thus indicated in **Figure A.8** by the portion of the curve labelled "Compressibility Included".

In general, the compressibility of liquids tends to be relatively low, and the pressure effect on B_o is thus not large. In this particular case, B_o decreases from 1.417 at the bubble point pressure to 1.389 at a pressure of 6,000 psia, which represents a volume decrease of only about 2% for a pressure increase of almost 50%. For some fluid systems, however, particularly lighter oils with relatively high GOR values, the effect can be significantly larger.

Gas Viscosity

Viscosity is a measure of resistance to flow of or through a medium. As a gas is heated, the molecules' movement increases and the probability that one gas molecule will interact with another increases. This translates into an increase in intermolecular activity and attractive forces. The viscosity of a gas is caused by a transfer of momentum between stationary and moving molecules. As temperature increases, molecules collide more often and transfer a greater amount of their momentum. This increases the viscosity.

You can select one of the following calculation methods to calculate the gas viscosity:

- Lee, Gonzalez and Eakin
- Carr, Kobayashi and Burrows (Dempsay version)
- Carr, Kobayashi and Burrows (Dranchuk version)

Live Oil Viscosity

Live oil viscosity is the measure of flow resistance of the live oil. Live oil refers to oil that is in equilibrium with any gas that may be present. If there is any free gas, the oil is also said to be saturated. If there is no free gas, but more could go into solution in the oil if it were present, the oil is said to be undersaturated.

You can select one of the following calculation methods to calculate the live oil viscosity:

- Chew and Connally
- Beggs and Robinson
- Khan

Undersaturated Oil Viscosity

For a given temperature, an oil is said to be undersaturated at any pressure above the bubble point pressure. Increasing the pressure would force more gas to go into solution if there was any, but above the bubble point pressure, there is no more free gas. With no more gas going into solution above the bubble point, the viscosity of the oil actually begins to increase with increasing pressure due to the compressibility of the oil. Since liquid compressibility is typically small, the effect of pressure on viscosity is much smaller above the bubble point than below.

A number of correlations have been proposed for computing the viscosity of undersaturated oils, and a few of these are described below. All of these procedures assume that the bubble point pressure is known at the temperature of interest, as well as the saturated oil viscosity corresponding to the bubble point pressure.

You can select one of the following calculation methods to compute the undersaturated oil viscosity:

- Vasquez and Beggs
- Beal
- Khan
- Abdul and Majeed

Dead Oil Viscosity Equation

The term Dead Oil refers to oil that has been taken to stock tank conditions and contains no dissolved gas (in other words, gas solution). Dead oil may exist at any pressure or temperature, but it is always assumed that all gas was removed at stock tank conditions. Any properties ascribed to a dead oil are thus characteristic of the oil itself.

Dead Oil Viscosity is the viscosity of an oil with no gas in solution. A number of the more useful methods for calculating this quantity are defined in the equations below.

The General Equation is defined as,

$$\mu_{do} = CEPT\left(\frac{100}{T}\right)^{SLP} \tag{A.17}$$

where:

μ_{do} *= dead oil dynamic viscosity, cP*

CEPT, SLP = constants for a given oil

T = oil temperature, °F

The ASTM Equation is defined as,

$$\log_{10}(\log_{10}Z) = A - B\log_{10}(T + 460) \tag{A.18}$$

where:

$Z = \nu_{do} + 0.7$

ν_{do} *= dead oil kinematic viscosity, cS*

A, B = constants for a given oil

T = oil temperature, °F

The kinematic viscosity, ν_{do} is given by,

$$\nu_{do} = \frac{\mu_{do}}{\rho_o} \tag{A.19}$$

where:

ρ_o *= density of the oil at the temperature of interest, expressed in g/cm3.*

The Eyring Equation is given by,

$$\nu_{do} = A\exp\left(\frac{1.8B}{T + 460}\right) \tag{A.20}$$

where:

A and B = constants for a given oil

Watson K Factor

You can choose to specify the Watson K Factor, or you can have HYSYS calculate the Watson K Factor. The default option is Specify.

The Watson K Factor is used to characterize crude oils and crude oil fractions. It is defined as,

$$K = \frac{T_B^{1/3}}{SG_o} \tag{A.21}$$

where:

K = Watson K factor

T_{TB} *= normal average boiling point for the crude oil or crude oil fraction, °R*

SG_o *= specific gravity of the crude oil or crude oil fraction*

For example, a particular kerosene cut, obtained over the boiling point range 284 - 482 °F, has a specific gravity of 0.7966. Then,

$$K = \frac{[0.5(284 + 482) + 460]^{1/3}}{0.7966} \tag{A.22}$$

$$= 11.86$$

Values of K typically range from about 11.5 to 12.4, although both lower and higher values are observed. In the absence of a known value, K = 11.9 represents a reasonable estimate.

Surface Tension

Surface tension is the measure of attraction between the surface molecules of a liquid. In porous medium systems (i.e. oil reservoirs), surface tension is an important parameter in the estimation of recoverable reserves because of its effect on residual saturations. On the other hand, most correlations and models for predicting two phase flow phenomena in pipelines are relatively insensitive to surface tension, and one can generally use an average value for calculation purposes. Calculations for wells have a somewhat stronger dependence on surface tension, in that this property can be important in predicting bubble and droplet sizes (maximum stable droplet size increases as surface tension increases), which in turn, can significantly influence the calculated pressure drop. Even then, however, surface tension typically appears in the equations raised to only about the ¼ power.

You can choose to have the surface tension calculated by HYSYS, or you can specify the surface tension. The default option is Calculate.

A.2 References

[1] Abbot, M. M., Kaufmann, T. G., and Domash, L., "A Correlation for Predicting Liquid Viscosities of Petro-leum Fractions", Can. J. Chem. Eng., Vol. 49, p. 379, June (1971).

[2] Abdul-Majeed, G. H., and Salman, N. H., "An Empirical Correlation for Oil FVF Prediction", J. Can. Petrol. Technol., Vol. 27, No. 6, p. 118, Nov.-Dec. (1988).

[3] Abdul-Majeed, G. H., Kattan, R. R., and Salman, N. H.,"New Correlation for Estimating the Viscosity of Under-saturated Crude Oils", J. Can. Petrol.Technol., Vol. 29, No. 3, p. 80, May-June (1990.)

[4] Al-Marhoun, M. A., "Pressure-Volume-Temperature Correlations for Saudi Crude Oils", paper No. SPE 13718, presented at the Middle East Oil Tech. Conf. and Exhib., Bahrain (1985)

[5] Al-Marhoun, M. A., "PVT Correlations for Middle East Crude Oils", J. Petrol. Technol., p. 660, May (1988).

[6] Al-Marhoun, M. A., "New Correlations for Formation Volume Factors of Oil and Gas Mixtures", J. Can. Petrol. Technol., Vol. 31, No. 3, p. 22 (1992).

[7] American Gas Association, "Compressibility and Supercompressibility for Natural Gas and Other Hydrocarbon Gases", Transmission Measurement Committee Report No. 8, December 15 (1985).

[8] American Petroleum Institute, API 44 Tables: Selected Values of Properties of Hydro-carbons and Related Compounds, (1975).

[9] Asgarpour, S., McLauchlin, L., Wong, D., and Cheung, V., "Pressure-Volume-Temperature Correlations for Wes-tern Canadian Gases and Oils", J. Can. Petrol. Technol., Vol. 28, No. 4, p. 103, Jul-Aug (1989).

[10] Baker, O., and Swerdloff, W., "Finding Surface Tension of Hydrocarbon Liquids", Oil and Gas J., p. 125, January 2 (1956).

[11] Beal, C., "The Viscosity of Air, Water, Natural Gas, Crude Oil and its Associated Gases at Oil Field Temperatures and Pressures", Trans. AIME, Vol. 165, p. 94 (1946).

[12] Beg, S. A., Amin, M. B., and Hussain, I., "Generalized Kinematic Viscosity-Temperature Correlation for Undefined Petroleum Fractions", The Chem. Eng. J., Vol. 38, p. 123 (1988).

[13]Beggs, H. D., and Robinson, J. R., "Estimating the Viscosity of Crude Oil Systems", J. Petrol. Technol., p. 1140, September (1975).

[14]Bradley, H.B. (Editor-in-Chief), Petroleum Engineering Handbook, Society of Petrol. Engrs (1987); Smith, H.V., and Arnold, K.E., Chapter 19 "Crude Oil Emulsions".

[15]Carr, N. L., Kobayashi, R., and Burrows, D. B., "Viscosity of Hydrocarbon Gases Under Pressure", Trans. AIME, Vol. 201, p. 264 (1954).

[16]Chew, J., and Connally, C. A., "A Viscosity Correlation for Gas Saturated Crude Oils", Trans. AIME, Vol. 216, p. 23 (1959).

[17]Dean, D. E., and Stiel, L. I., "The Viscosity of Nonpolar Gas Mixtures at Moderate and High Pressures", AIChE J., Vol. 11, p. 526 (1965).

[18]Dempsey, J. R., "Computer Routine Treats Gas Viscosity as a Variable", Oil and Gas J., p. 141, August 16 (1965).

[19]Dokla, M. E., and Osman, M. E., "Correlation of PVT Properties for UAE Crudes", SPE Form. Eval., p. 41, Mar. (1992).

[20]Dranchuk, P.M., Purvis, R.A., and Robinson, D.B., "Computer Calculations of Natural Gas Compressibility Factors Using the Standing and Katz Correlations", Inst. of Petrol. Technical Series, No. IP74-008, p. 1 (1974).

[21]Dranchuk, P. M., and Abou-Kassem, J. H., "Calculations of Z Factors for Natural Gases Using Equa-tions of State", J. Can. Petrol. Technol., p. 34, July-Sept. (1975).

[22]Dranchuk, P. M., Islam, R. M. , and Bentsen, R. G., "A Mathematical Representation of the Carr, Kobayashi, and Burrows Natural Gas Viscosity Cor-relations", J. Can. Petrol. Technol., p. 51, January (1986).

[23]Elsharkawy, A. M., Hashem, Y. S., and Alikan, A. A., Compressibility Factor for Gas-Condensates", Paper SPE 59702, presented at the SPE Permian Basin Oil and Gas Recovery Conf., Midland, TX, March (2000).

[24]Eyring, H., "Viscosity, Plasticity and Diffusion as Examples of Absolute Reaction Rates", J. Chem. Phys., Vol. 4, p. 283 (1936).

[25]Gas Processors Association, Engineering Data Book, Tulsa, Oklahoma, 9th Edition (1977), 10th Edition (1987).

[26]Glasø, Ø., "Generalized Pressure-Volume-Temperature Correlations", J. Petrol. Technol., p. 785, May (1980).

[27]Gomez, J. V., "Method Predicts Surface Tension of Petroleum Fractions", Oil and Gas J., p. 68, December 7 (1987).

[28]Gray, H. E., "Vertical Flow Correlation - Gas Wells", API Manual 14 BM, Second Edition, Appendix B, p. 38, American Petroleum Institute, Dallas, Texas, January (1978).

[29]Gregory, G. A., "Viscosity of Heavy Oil/Condensate Blends", Technical Note No. 6,

[30]Neotechnology Consultants Ltd., Calgary, Canada, July (1985).

[31]Gregory, G. A., "Pipeline Calculations for Foaming Crude Oils and Crude Oil-Water Emulsions", Technical Note No. 11, Neotechnology Consultants Ltd., Calgary, Canada, January (1990).

[32]Gregory, G. A., "Calculate the Density of Non-hydrocarbon Gases Correctly", Technical Note No. 24, Neotechnology Consultants Ltd., Calgary, Canada, November (2000).

[33]Guth, E., and Simha, R., Kolloid-Zeitschrift, Vol. 74, p. 266 (1936).

[34]Hatschek, E., "Die Viskositat der Dispersoide", Kolloid-Zeitschrift, Vol. 8, p. 34 (1911).

[35]Hougen, O. A., Watson, K. M., and Ragatz, R. A., Chemical Process Principles, Vol. 2, p. 593, John Wiley & Sons, Inc., New York, N.Y. (1959).

[36]Jossi, J. A., Stiel, L. I., and Thodos, G., "The Viscosity of Pure Substances in the Dense, Gaseous, and Liquid Phases", AIChE J., Vol. 8, p. 59 (1962).

[37]Katz, D. L., and Firoozabadi, A., "Predicting Phase Behaviour of Condensate/Crude Oil Systems Using Methane Interaction Coefficients", J. Petrol. Technol., p. 1649, November (1978).

[38]Kay, W. B., "Density of Hydrocarbon Gases and Vapor at High Temperature and Pressure", Ind. Eng. Chem., p. 1014, September (1936).

[39]Khan, S. A., Al-Marhoun, M. A., Duffuaa, S. O., and Abu-Khamsin, S. A., "Viscosity Correlations for Saudi Arabian Crude Oils", paper No. SPE 15720, presented at the 5th SPE Middle East Oil Show, Manama, Bahrain, March (1987).

[40]Lasater, J. A., "Bubble Point Pressure Correlation", Trans. AIME, Vol. 213, p. 379, (1958).

[41]Lee, A. L., Gonzalez, M. H., and Eakin, B. E., "The Viscosity of Natural Gases", J. Petrol. Technol., Vol. 18, p. 997 (1966).

[42]Manning, R. E., "Computation Aids for Kinematic Viscosity Conversions from 100 and 210 oF to 40 and 100 oC", J. of Testing and Evaluations (JVETA), Vol. 2, p. 522, November (1974).

[43]Meehan, D. N., "A Correlation for Water Viscosity", Petrol. Eng. Int., July (1980).

[44]McCain, W. D., "Black Oils and Volatile Oils - What's the Difference?", Pet. Eng. Intl., p. 24, November (1993).

[45]McCain, W. D., "Volatile Oils and Retrograde Gases - What's the Difference?", Pet. Eng. Int., p. 35, January (1994a).

[46]McCain, W. D., "Heavy Components Control Reservoir Fluid Behaviour", J. Petrol. Technol., p. 764, September (1994).

[47]Moses, P. L., "Engineering Applications of Phase Behaviour of Crude Oil and Condensate Systems", J. Petrol. Technol., p. 715, July (1986).

[48]Ng, J. T. H., and Egbogah, E. O., "An Improved Temperature-Viscosity Correlation for Crude Oil Systems", Paper No. 83-34-32, presented at the 34th Ann. Tech. Mtg. of The Petrol. Soc. of CIM, Banff, Alta, May (1983).

[49]Petrosky, G. E., and Farshad, F. F., "Pressure-Volume-Temperature Correlations for Gulf of Mexico Crude Oils", Paper No. SPE 26644, presented at the 68th Ann. Tech. Conf. & Exhib. of the SPE, Dallas, TX, Sept. (1987).

[50]Reid, R. C., Prausnitz, J. M., and Sherwood, T. K., The Properties of Gases and Liquids, 3rd Edition, McGraw-Hill Book Co., New York (1977).

[51]Riazi, M. R., and Daubert, T. E., "Simplify Property Predictions", Hydrocarbon Processing, p. 115, March (1980).

[52]Shu, W. R., "A Viscosity Correlation for Mixtures of Heavy Oil, Bitumen, and Petroleum Fractions", SPE Jour., p 277, June (1984).

[53]Society of Petroleum Engineers, Petroleum Engineering Handbook, Chapter 19, "Crude Oil Emulsions", by Smith, H.V., and Arnold, K.E., p. 19-6, Richardson, Texas (1987).

[54]Society of Petroleum Engineers, Petroleum Engineering Handbook, H.B. Bradley, Editor-in Chief, Richardson, Texas (1987).

[55]Standing, M. B., "A Pressure-Volume-Temperature Correlations for Mixtures of California Oils and Gases", Drill. Prod. Practice, API, p. 247 (1947).

[56]Standing, M. B., Volumetric and Phase Behaviour of Oil Field Hydrocarbon Systems, Society of Petroleum Engineers of AIME, Dallas, Texas, 8th Printing (1977).

[57]Standing, M. B., and Katz, D. L., "Density of Natural Gases", Trans. AIME, Vol. 146, p. 140 (1942).

[58]Sutton, R. P., "Compressibility Factor for High Molecular Weight Reservoir Gases", Paper SPE 14265, presented at the Ann. Tech. Mtg. and Exhib. of the SPE, Las Vegas, September (1985).

[59]Sutton, R. P., and Farshad, F., "Evaluation of Empirically Derived PVT Properties for Gulf of Mexico Crude Oils", SPE Res. Eng., p. 79, Feb. (1990).

[60]Twu, C. H., "Generalized Method for Predicting Viscosities of Petroleum Fractions", AIChE J., Vol. 32, No. 12, p. 2091 (1986).

[61]Twu, C. H., and Bulls, J. W., "Viscosity Blending Tested", Hydrocarbon Proc., p. 217, April (1981).

[62]Vasquez, M., and Beggs, H. D., "Correlations for Fluid Physical Property Prediction", Paper SPE 6719, presented at the 52nd Annual Technical Conference and Exhibition, Denver, Col. (1977), Published in J. Petrol. Technol., p. 968 (1980).

[63]Watson, K. M., and Nelson, E. F., "Improved Methods for Approximating Critical and Thermal Properties of Petroleum Fractions", Ind. Eng. Chem., Vol. 25, p. 880, August (1933).

[64]Wichert, E., and Aziz, K., Compressibility Factor of Sour Natural Gases", Can. J. Chem. Eng., Vol. 49, p. 267, April (1971).

[65]Wichert, E., and Aziz, K., "Calculated Z's for Sour Gases", Hydrocarbons Processing, p. 119, May (1972).

[66]Woelflin, W., "Viscosity of Crude Oil Emulsions", Oil and Gas J., Vol. 40, No. 45, p. 35, March 19 (1942).

B Black Oil Transition Methods

B.1Transition Methods

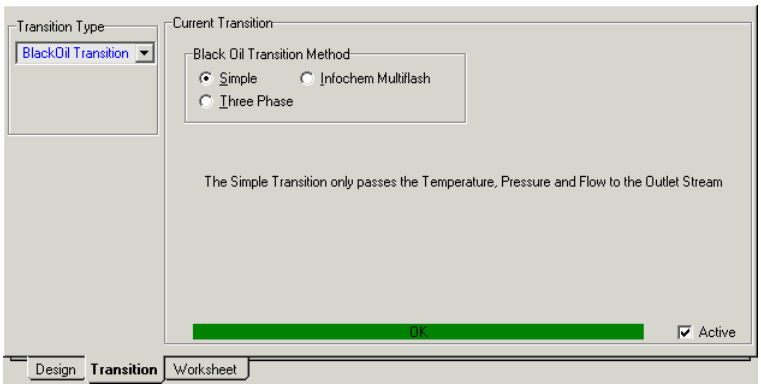
The Black Oil Transition is the engine that is used in translating between black oils and compositional models. There are two available methods for Black Oil Transition:

- Simple
- Three Phase

B.1.1 Simple Method

The Simple method is a set of black oil transitions that do not require any additional user input at the operation level. The Simple method can be used when transitioning between the types of streams described in the following sections.

Figure B.1



Black Oil to Black Oil

If two different black oil fluid packages are available within a flowsheet, the Simple method can be used for the transitioning between them. In this case, you need to specify the viscosity method and any corresponding viscosity data (such as a viscosity curve if required for that particular method) on the outlet stream. The transferrable properties include Temperature, Pressure, Phase Mass Flow Rates, Watson K (if

necessary), Surface Tension (if necessary), Oil Gravity, Gas Gravity or Gas Composition, and Water Gravity.

Compositional to Black Oil

When using a traditional compositional fluid package and a black oil fluid package in the same flowsheet, it maybe desirable to set the inlet as the compositional stream and the outlet as the black oil stream. This way, the Simple method can be used. The Temperature, Pressure, Phase Mass Flow Rates, Watson K (if necessary), Surface Tension (if necessary), Oil Gravity, Gas Gravity or Gas Composition, and Water Gravity are all transferred to the outlet.

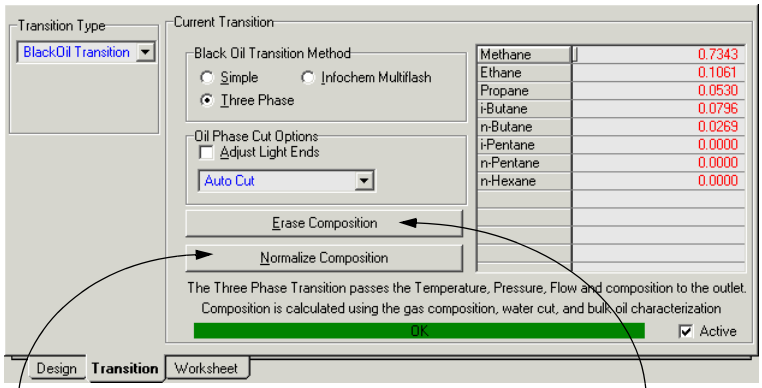
Black Oil to Compositional

The Simple method can also be used to convert between black oil and compositional material streams. In this situation the Temperature, Pressure, and Overall Mass Flow Rate are transferred to the outlet compositional material stream from the black oil inlet stream. As such, the outlet stream must already have a defined composition. This feature is primarily useful in providing flowsheet continuity. For a more thorough form of transition the Three Phase method should be used.

B.1.2 Three Phase Method

The Three Phase method of transition is used when thorough conversion from black oil to compositional transition is required. The Three Phase method independently characterizes the three phases of the black oil as compositional phases and then recombines the phases back into a single compositional stream.

Figure B.2



The **Normalize Composition** button is useful when many components are available, but you want to specify compositions for only a few. When you enter the compositions, click the **Normalize Composition** button and HYSYS ensures the Total is **1.0**, while also specifying any **<empty>** compositions as zero. If compositions are left as **<empty>**, HYSYS cannot perform the flash calculation on the stream.

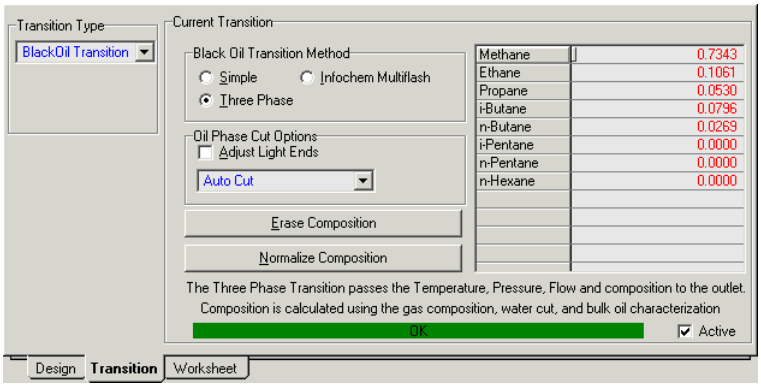
Gas Phase

The black oil gas phase is converted to a compositional model by relying on user inputs. If the black oil inlet stream has a known gas phase composition it is used by the Black Oil Transition. The composition as displayed in **Figure B.2** is referred to as the Operating Gas Composition. You can overwrite the Operating Gas Composition. If you do not modify the Operating Gas Composition it will automatically reflect any changes that occur to the inlet black oil stream gas composition. If you modify the Operating Gas Composition, any changes to the inlet stream black oil gas composition will not be propagated to the Operating Gas Composition.

Oil Phase

The Black Oil oil phase is transitioned to a compositional model using the HYSYS Oil Manager Bulk Properties Assay Definition methods. The transition passes on the oil phase standard density and oil phase Watson K to the Oil Manager and an appropriate assay and blend is created for the user. This functionality is automatically done by HYSYS and no additional user interaction is required.

Figure B.3



You can use the Oil Phase Cut Options to adjust the light end and auto-characterize the hypocomponents into user specified light end components. You have the option to have the HYSYS Oil Manager perform an auto cut, or specify the appropriate number of cuts.

Water Phase

The black oil water phase is assumed to be pure water by the black oil transition.

After the transition models the three phases, it recombines them into the outlet stream and passes on Temperature, Pressure, and Overall Mass Flow Rate. At this point, the outlet stream will flash and in most cases three corresponding compositional phases will be calculated. The outlet Vapour

phase represents the inlet Gas phase, the Liquid phase represents the Oil phase, and the Aqueous phase represents the Water phase. The outlet stream's property package and the flash determine the existence of phases, the phase fractions, and the phase properties. In most cases these items will be very similar to the inlet black oil stream particularly when using a property package such as Peng-Robinson. Overall Mass balance between the inlet and outlet is assured.

B.1.3 Infochem Multiflash

Multiflash is an advanced software package for modeling the properties of gases, liquids and solids. It consists of a comprehensive library of thermodynamic and transport property models, a physical property databank, methods for characterising and matching the properties of petroleum fluids and multiphase flashes capable of handling any combination of phases.

Refer to **Chapter 2 - Multiflash for HYSYS Upstream** for more information on Infochem Multiflash.

2 Multiflash for HYSYS Upstream

2.1 Introduction

Multiflash is an advanced software package for modeling the properties of gases, liquids and solids. It consists of a comprehensive library of thermodynamic and transport property models, a physical property databank, methods for characterising and matching the properties of petroleum fluids and multiphase flashes capable of handling any combination of phases.

This chapter describes the use of Multiflash with HYSYS Upstream (a product of Aspen Technology Inc.). Multiflash is available as a property package in the COMThermo thermodynamics option. The use of the Multiflash GUI for Microsoft Windows is described in the **User Guide for Multiflash for Windows** which is supplied as a pdf file (in the Multiflash installation directory) and as online help.

2.1.1 Installing Multiflash

Multiflash for COMThermo is installed as part of the HYSYS Upstream. The installation instructions in the **User Guide for Multiflash for Windows** refer to the standalone version of Multiflash and do not apply to HYSYS Upstream.

2.2 Multiflash Property Package

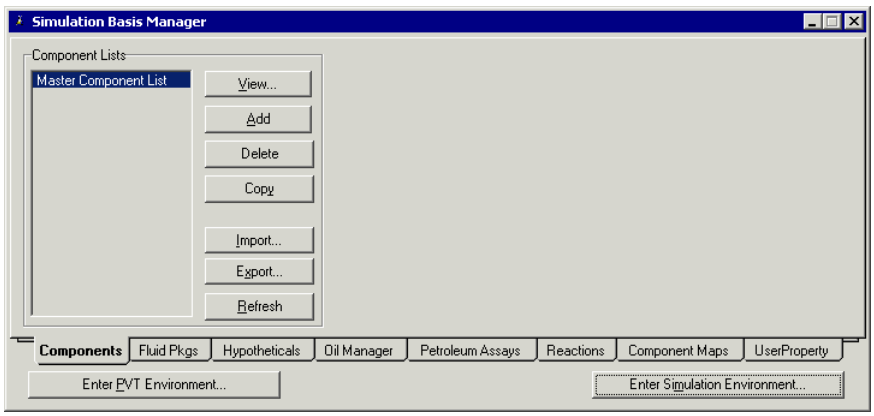
A Multiflash property package consists of a set of components and thermodynamic and transport property models for several phases. In HYSYS terminology it corresponds to a Fluid Package plus a set of components. You may create several different property packages with different components and/or models as required.

2.2.1 Adding a Multiflash Property Package

To add a Multiflash property package to a new case, you must start HYSYS.

1. From the **File** menu, select **New** and then **Case**.
   The Simulation Basis Manager appears.

Figure 2.1

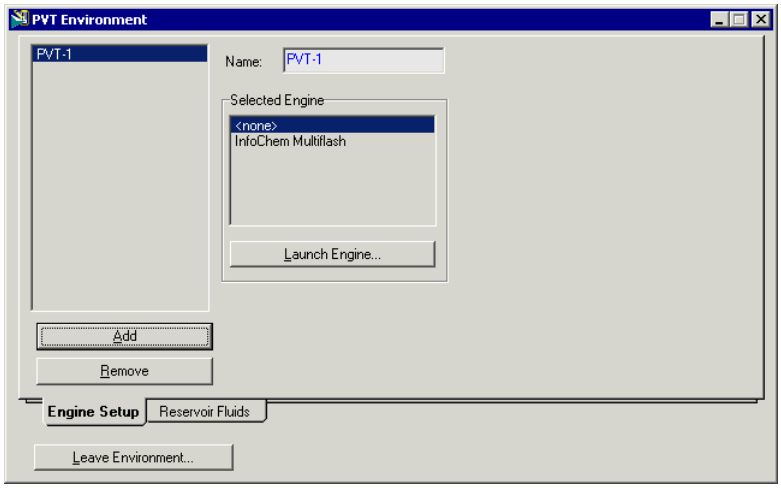


The components for the property package must be selected within the Multiflash GUI, for more information refer to the section on Components. You cannot select the components within HYSYS.

Refer to **Chapter 8 - PVT Environment** in the **HYSYS Simulation Basis** guide for more information on the PVT Environment Manager.

2. Click on the **Enter PVT Environment** button. The PVT Environment Manager appears.
3. On the **Engine Setup** tab, click the **Add** button. A new PVT package (PVT-1) is added to the PVT package list.

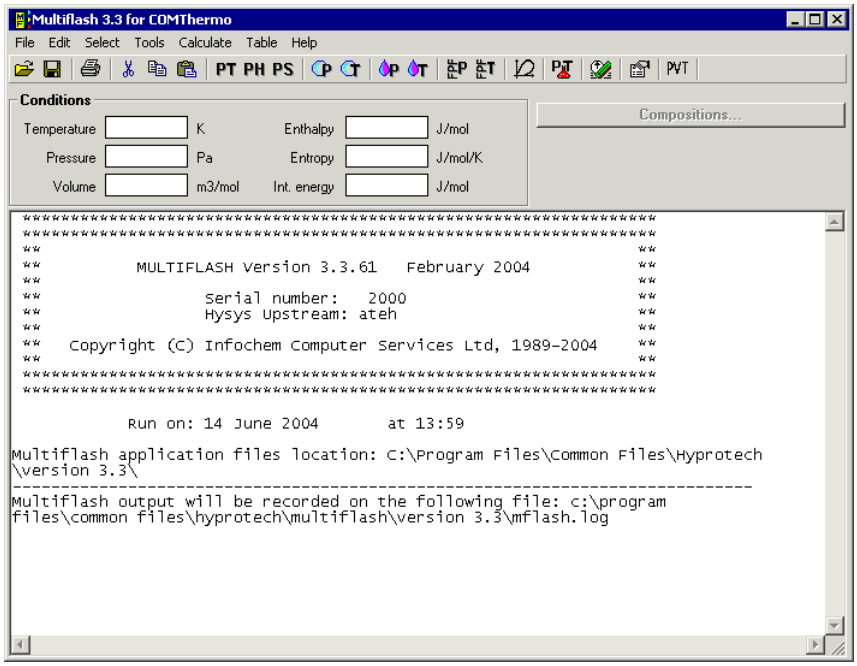
Figure 2.2



4. From the **Selected Engine** list, select **Infochem Multiflash**.

5. Click the **Launch Engine** button to launch the Infochem Multiflash GUI.

Figure 2.3



Multiflash Flash

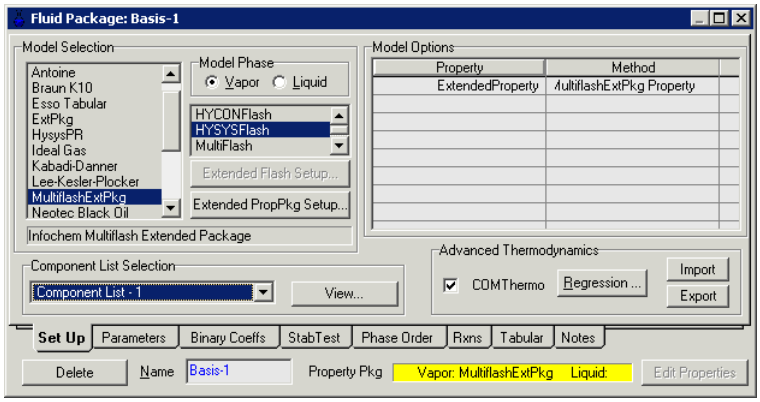
The Multiflash flash is very reliable in most circumstances, including situations with multiple liquid phases and solids. The convergence tolerance is very tight and, therefore, it may take longer to converge than some other options but the result is correct. For simple vapour-liquid equilibrium calculations any flash method should be reliable.

The Multiflash flash is capable of doing calculations for solid, liquid and gas phases but for HYSYS Upstream only the gas and liquid phases are used.

HYSYS Flash

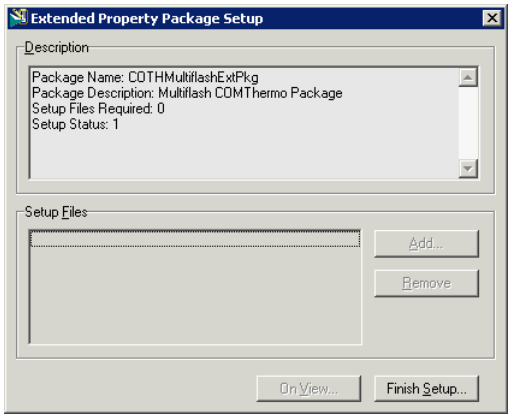
If you are using the HYSYS flash and you want to change the flash settings, you can click the Advanced button on the HYSYS-Multiflash Setup view, which opens the Fluid Package view.

Figure 2.4



When you click the Extended PropPkg Setup button, the Extended Property Package Setup view appears.

Figure 2.5



Refer to **Section 2.2.2 - Configuring a Property Package** for information on configuring the property package.

Clicking the Finish Setup button opens the Multiflash GUI.

2.2.2 Configuring a Property Package

The Multiflash Windows program (GUI) is used to configure a property package for HYSYS Upstream but it can also be used to carry out flash calculations, to plot phase envelopes or to regress model parameters to match experimental measurements. The use of the GUI is described in the **User Guide for Multiflash for Windows**. This section gives a brief description of how it is used with HYSYS and highlights extra features provided for the HYSYS setup.

To configure a Property Package it is necessary to specify the following information:

- The **components** (substances) that are included. These maybe any components from databanks supported by Multiflash or petroleum fraction pseudocomponents.
- The thermodynamic/physical property **models** that will be used to evaluate properties such as volume, enthalpy, fugacity coefficients, transport properties.
- The **binary interaction parameters** (BIPs) that will be used by the model(s).
- The **phases** which may be included.

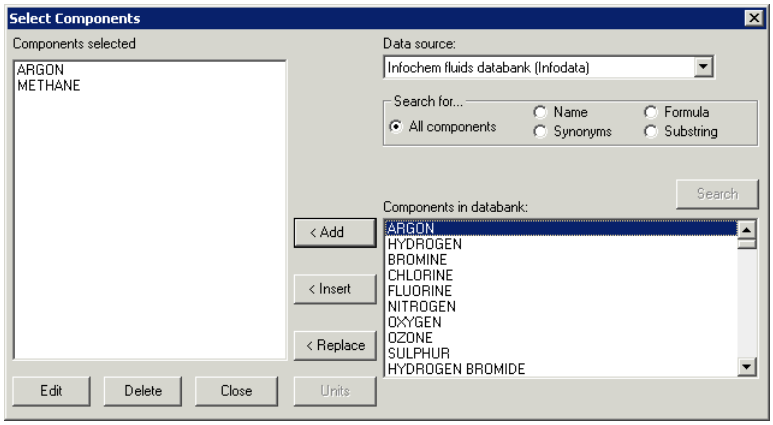
Components

The Select/Components menu item allows you to specify the components in the property package. Components may be selected from a databank by clicking on a component name or typing the name of a component. Components in a databank may be searched for by name, part of a name or by formula as shown in the figure below.

Petroleum fraction pseudocomponents may be specified by entering properties such as molecular weight, specific gravity, and Multiflash will then estimate any other properties required.

In addition there is an option to define a new component by entering in all the data.

Figure 2.6

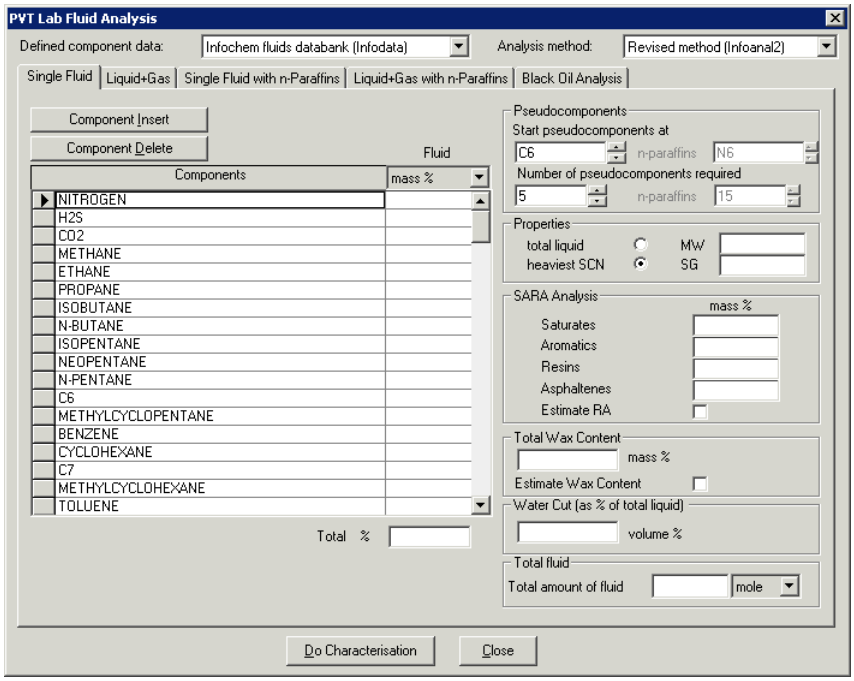


Petroleum Fluid Characterisation

An alternative way of entering compositional information is to use the PVT Lab Input item on the Select menu. This displays a form that allows input of the typical information included in a PVT report produced by oil industry service companies. The representation of the fluid can be controlled by the user by selecting the number of pseudocomponents used to describe the fluid is and how to group or split pseudocomponents.

Other information used to set up solid deposition models in Multiflash may also be entered.

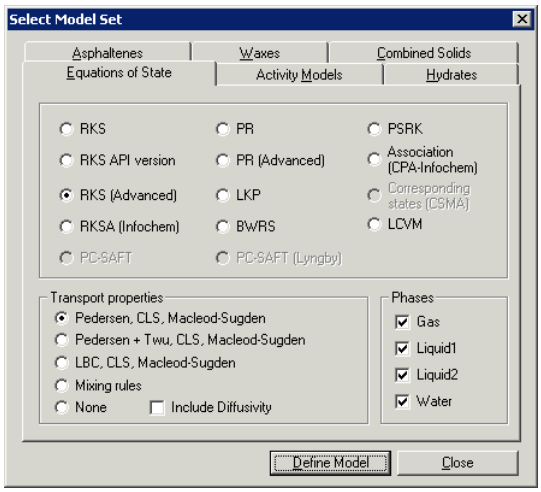
Figure 2.7



Physical Property Models and Phases

Multiflash offers a wide choice of models for representing the properties of fluid and solid phases. Models are defined using the Select/Model Set menu item.

Figure 2.8



The Select Model Set window has several tabs that group together different types of models. **Figure 2.8** shows the equation of state models. A number of different transport property options may be selected along with the thermodynamic models. Models that are not part of your license will be greyed out and cannot be selected. After choosing a model click the Define Model button and then Close to return to the main window.

The number of phases available in a Multiflash property package can be controlled by the checkboxes in the lower right hand corner of the Select Model Set window. By default most of the equation of state models are set up for four phases: Gas, Liquid1, Liquid2 and Water (aqueous phase). In cases where two liquid phases in addition to water are unlikely performance can be improved by clearing the **Liquid2** checkbox. Similarly if an aqueous phase will not be present the Water phase may be unchecked.

Although Multiflash does not have restrictions on the type and number of phases, the HYSYS flash or HYSYS unit operations may not support all the phases.

Binary Interaction Parameters

Most models require values of binary interaction parameters (BIPs) to make sure the model represents the interactions between components in a mixture. BIPs for the equation of state models are mostly generated automatically using correlations. For activity models a large number of BIPs are stored on the supplied BIP databanks but it is necessary for the user to enter any missing values. This is done using the Tools/BIPs menu item.

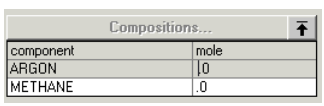
2.2.3 Carrying Out Calculations

Once the components, models and other parameters have been set up as described in **Section 2.2.2 - Configuring a Property Package**, you may use the Multiflash GUI to carry out property calculations.

Composition

To enter a mixture composition click on the Compositions button. The amount of each component can be entered in the mole field. The amounts are total moles or mass rather than mole fractions or mass fractions.

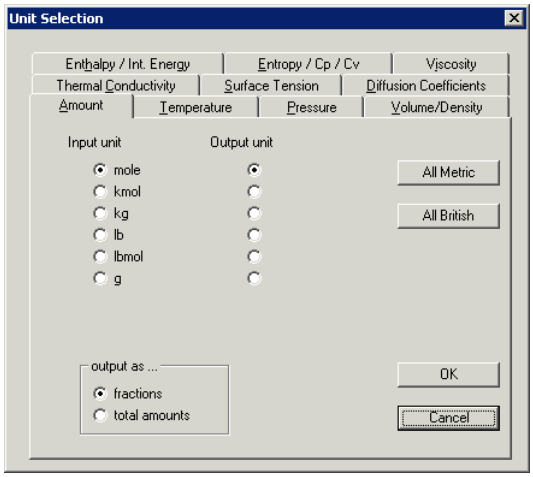
Figure 2.9



Units

Units of measurement for the Multiflash GUI can be set using the Select/Units menu item. This only affects operation of the GUI and has no connection with units in HYSYS.

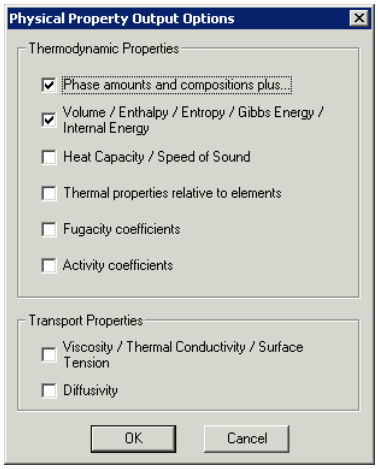
Figure 2.10



Flash Calculations

To carry out a flash calculation set the values of known quantities in the Input Conditions display and select an option from the Calculate menu. There are toolbar buttons for many of the common flash calculations. Calculated properties are displayed in the main window. The properties calculated depend on the property output level selected. This can be changed using the Select/Property Output menu item.

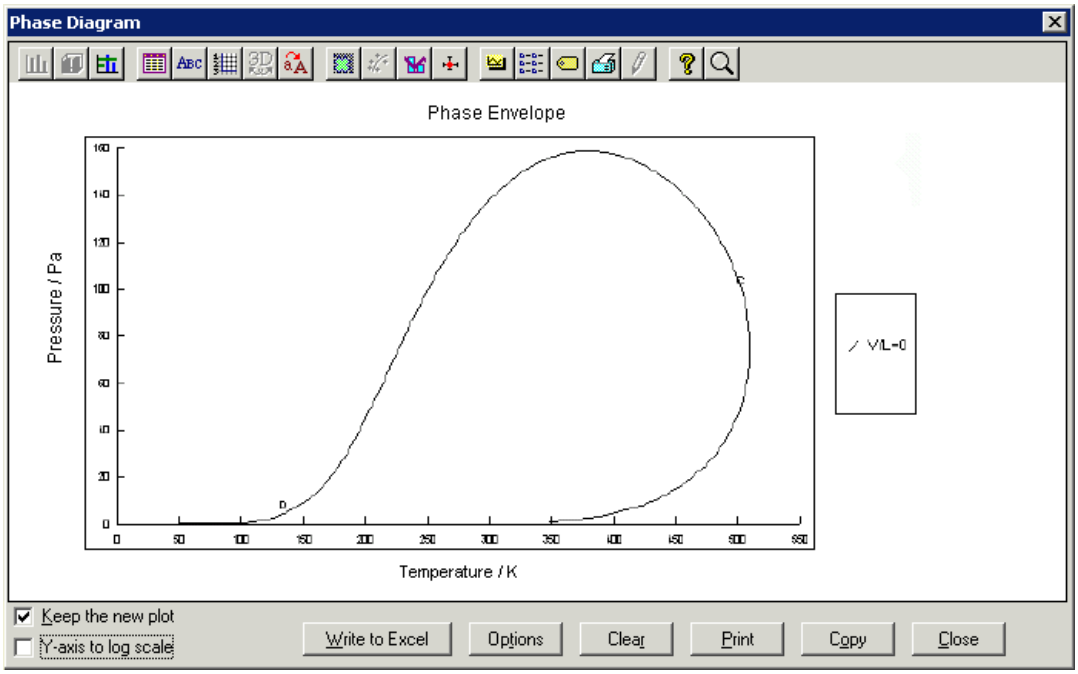
Figure 2.11



Phase Envelope

Phase boundaries and lines of fixed volume, enthalpy, entropy can be plotted. Use the Calculate/Phase Envelope menu item. To plot a vapour-liquid phase boundary click on the VLE AutoPlot button – see example output below. Other types of diagrams may be plotted by setting the Basis and X/Y axes as required.

Figure 2.12



Matching Experimental Data

It is possible to adjust models to match measured values of quantities such as dew points, bubble points and viscosities for petroleum fluids. The mixture must include petroleum fraction pseudocomponents for the matching options to work. To match data use the Tools/Matching menu item.

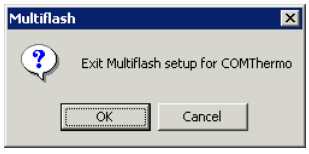
Online Help

Help is provided by two items on the Help menu. Help Topics is an online version of the Multiflash for Windows User Guide and Multiflash Error Codes gives an explanation of any error or warning messages issued by Multiflash.

Returning to HYSYS

After a property package has been configured you may return to HYSYS by selecting the Exit item from the File menu. This will display a warning message:

Figure 2.13



Click OK to return to HYSYS, or Cancel to return to Multiflash to allow further editing of the configuration.

3 Lumper and Delumper

3.1 Lumper

The Lumper is an upstream operation, which allows the user to blend multiple (well) streams to one stream with a reduced number of components to the total components into the unit operation. This technique is used to map petroleum and process thermodynamic properties together.

The Lumper unit operation is currently available only in steady state mode. The user can attach one or more streams using one or more different fluid packages with a total of *n* distinct components. In the simulation environment, the user then adds a new equation of state (EOS) fluid package, fluid package name and components list name to lump the *n* distinct components into *y* components using the Montel and Gouel[1] or Custom lumping methods.

3.1.1 Lumper Property View

You can also add the Lumper by clicking the F12 hot key.

You can also open the Object Palette by clicking the F4 hot key.

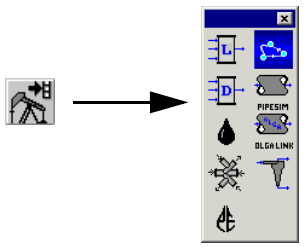
There are two ways that you can add a Lumper to your simulation:

1. From the **Flowsheet** menu, click **Add Operation**. The UnitOps view appears.
2. Click the **Upstream Ops** radio button.
3. From the list of available unit operations, select **Lumper**.
4. Click the **Add** button.

OR

1. From the **Flowsheet** menu, click **Palette** (or press **F4**). The Object Palette appears.
2. Click on the **Upstream Ops** icon. The Upstream Object Palette appears.

Figure 3.1

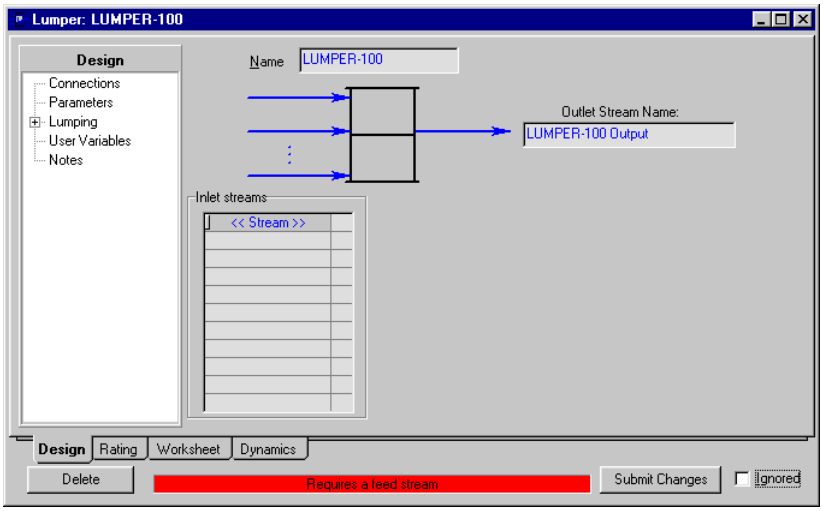


Lumper icon

3. In the Upstream Object Palette, double-click the **Lumper** icon.

The Lumper property view appears.

Figure 3.2



You can also delete a Lumper by clicking on the Lumper icon on the PFD and pressing the **DELETE** key.

To delete the Lumper operation, click the **Delete** button. HYSYS will ask you to confirm the deletion.

To ignore the Lumper during calculations, activate the Ignored checkbox. HYSYS completely disregards the operation (and

cannot calculate the outlet stream) until you restore it to an active state by deactivating the checkbox.

3.1.2 Design Tab

The Design tab consists of the following pages:

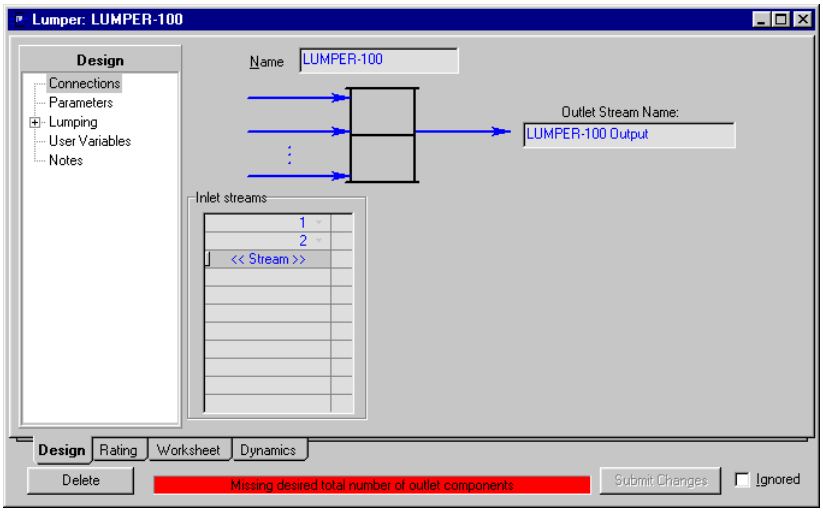
- Connections
- Parameters
- Lumping
- User Variables
- Notes

Connections Page

You can specify one or more inlet streams using one or more different fluid packages. The fluid packages may have different component lists.

The Connections page is used to define all of the connections to the Lumper. You can specify the inlet streams to attach to the operation. The name of the operation can be changed in the Name field, and you can also change the name of the outlet stream in the Outlet Stream Name field.

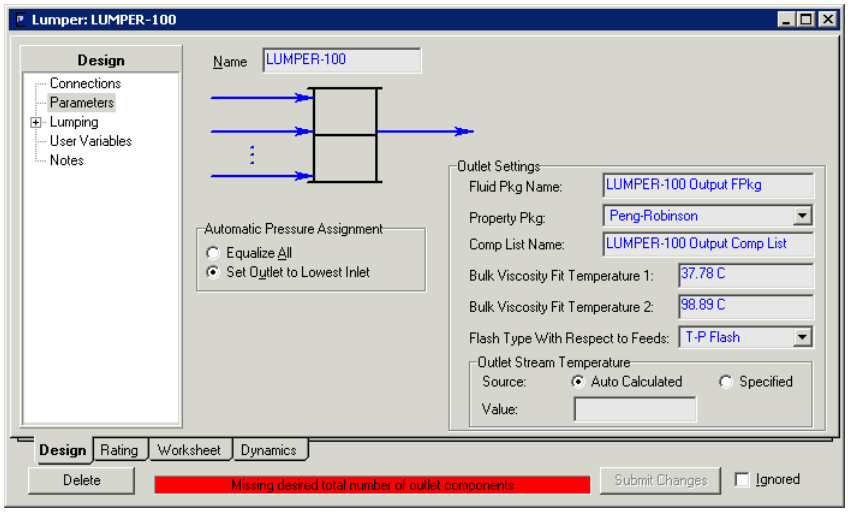
Figure 3.3



Parameters Page

The Parameters page allows you to specify the outlet settings and automatic pressure assignment.

Figure 3.4



The Outlet Settings group consists of the following parameters:

Parameters	Description
Fluid Pkg Name	Allows you to specify the name for the fluid package to be generated.
Property Pkg	You can select the Equation of State (EOS) property package from the drop-down list: • Peng-Robinson • PRSV • Sour PR • SRK • Kabadi-Danner • Sour SRK A new fluid package with components will be created during the delumping process.
Comp List Name	Allows you to specify the name for the component list to be generated.
Bulk Viscosity Fit Temperature 1	The first temperature at which the outlet stream bulk viscosity is to match that of the combined feed.

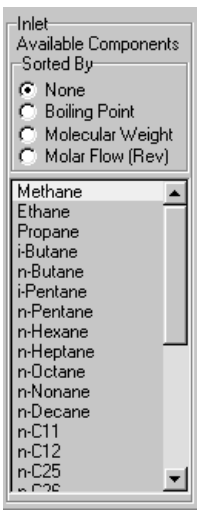
For more information on the EOS property packages, refer to the section on the **Equations of State (EOS)** in the **HYSYS Simulation Basis** guide.

Parameters	Description
Bulk Viscosity Fit Temperature 2	The second temperature at which the outlet stream bulk viscosity is to match that of the combined feed.
Flash Type with Respect to Feeds	You can select the flash type to be used for the outlet stream with respect to the combined feed. There are two options: • T-P Flash • P-H Flash
Outlet Stream Temperature Group	This group appears when you select T-P as the flash type from the Flash Type with Respect to Feeds drop-down list. There are two radio buttons in the Outlet Stream Temperature group: • **Auto Calculated**. The outlet stream temperature will be automatically calculated based on the combined feed. You cannot specify the outlet stream temperature in the Value field. • **Specified**. You have to specify the outlet stream temperature in the Value field.

The Automatic Pressure Assignment group consists of the following radio buttons:

- **Equalize All**. Click this radio button, if you want to force all stream pressure to be the same.
- **Set Outlet to Lowest Inlet**. Click this radio button, if you want the outlet pressure to be the lowest inlet pressure.

Lumping Page



Inlet group

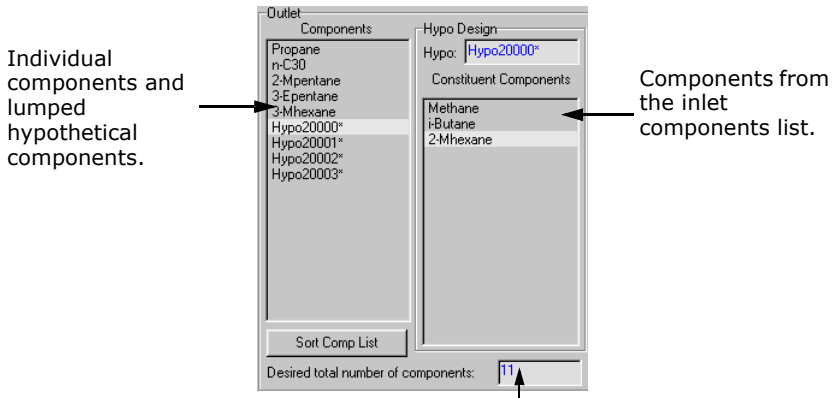
On the Lumping page, you can select the lumping method that you want to use. The outlet component list shown on the Lumping page can be constructed by a combination of the following three ways:

- **Keeping inlet components**: You can select any number of inlet components to keep as individual components in the outlet component list. You can also click the **Keep All Comps** button to keep all the inlet components.
- **Manually creating user hypothetical components (User Hypos**): After creating a hypothetical group, you can create any number of user hypos and decide the constituent inlet components for each user hypo. Any empty user hypo will be deleted from the outlet component list when you submit the changes or close the Lumper property view.

- **Automatically creating hypothetical components (Auto Hypos)**: When the desired total number of outlet components is more than the sum of the kept components and non-empty user hypos, the difference will be the number of auto hypos to be generated automatically by the lumping method you have selected. You can select a lumping method by expanding the Lumping page of the Design tab. The default method is Montel and Gouel. Once an auto hypo is generated, you are allowed to empty or delete it, but not to change the contained components.

As a result, the outlet component list may contain three groups of components: directly kept components, user created hypos, and automatically generated hypos. The outlet components are first ordered by groups in the aforementioned order and then, within the first two groups, can be sorted by clicking the Sort Comp List button. The auto hypos cannot be sorted since they are generated on the fly.

Figure 3.5

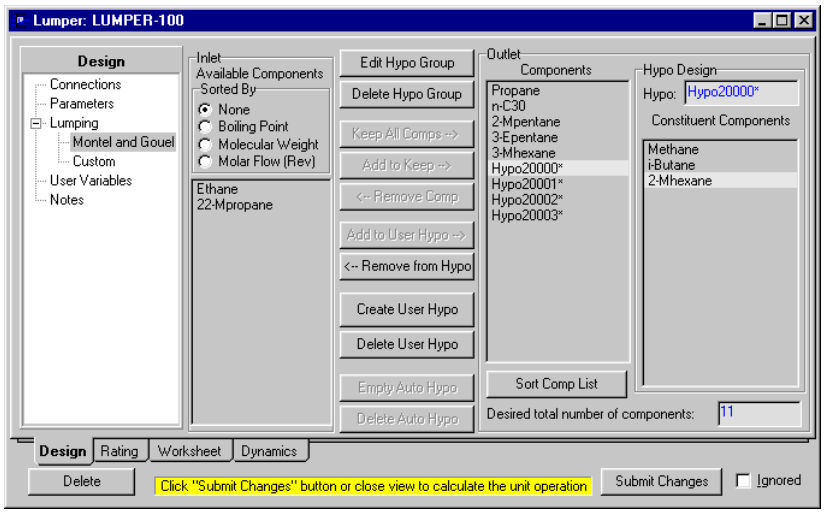


You can specify the total number of outlet components in the Desired total number of components field. The valid range for this number depends on the current state of the outlet components list. The minimum number equals the total number of kept components and the non-empty user hypos plus one (if there is one or more inlet components left). The maximum number equals the total number of kept components and the non-empty user hypos plus the remaining number of inlet components. If you enter an invalid number in this field, the trace window will show you the valid range.

There are currently three lumping methods:

- **Montel and Gouel**[1]. The Montel and Gouel method is the default lumping method. This method uses an iterative clustering algorithm around mobile centers, resulting in a classification into hypothetical components optimum with respect to the considered equation of state (EOS) and the chosen lumping properties. This method is therefore only applicable to the EOS property package

Figure 3.6



For the Montel and Gouel method, both the lumping properties and their weights are fixed as follows:

Lumping Property	Weighting Factor
EOS sqrt (a)	1.0
EOS b	1.0
EOS m	0.1
Mw	0.5

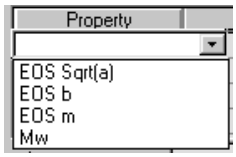
where:

EOS means the equation of state such as PR and SRK

sqrt (a), b and m are the square root of a, b and m respectively used in the EOS

Mw is the molecular weight of a component

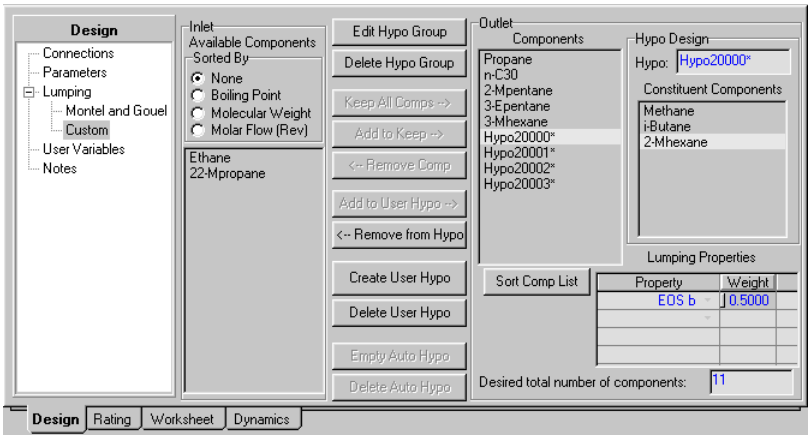
You must add at least one lumping property.



The weighting factors must be between 0 and 1.

- **Custom**. The Custom method uses the same iterative clustering algorithm as the Montel and Gouel method, although you can select lumping properties and specify weighting factors to be used in the lumping algorithm.

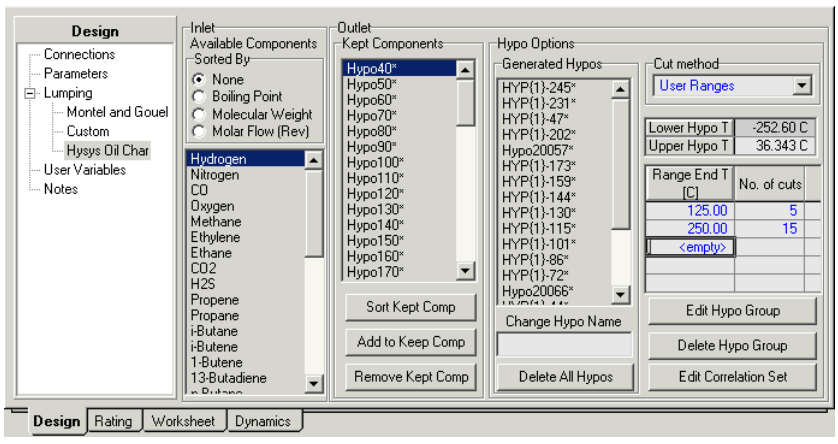
Figure 3.7



Refer to **Section 4.2 - Oil Characterization** from the **HYSYS Simulation Basis** guide for more information.

- **HYSYS Oil Characterization**. The HYSYS Oil Characterization method uses the combined feed to construct the working curves similar to those used in the blend in the oil characterization. The working curves then are cut according to the selected cutting method which are: Auto Cut, User Ranges, or User Num of Cuts (in other words, exactly the same as in the blend).

Figure 3.8



When the HYSYS Oil Characterization method is used in lumping, the generated hypothetical components do not have the knowledge on the distribution of the inlet components. Therefore, if the inlet components are first lumped using this method, the delumping will not be able to recover the original inlet components, except the ones kept in the lumper's outlet component list.

Sorting the Inlet Components

You can sort the available components in the inlet group by clicking on the appropriate radio button.

Radio Button	Description
None	The components remain in the original order. This is the order when you added the components to the fluid package component list.
Boiling Point	You can sort the components according to the boiling point of each component.
Molecular Weight	You can sort the components according to the molecular weight of each component.
Molar Flow (Rev)	You can sort the components according to the molar flow of each component. The component with the largest molar flow is on the top.

Creating a Hypothetical Group

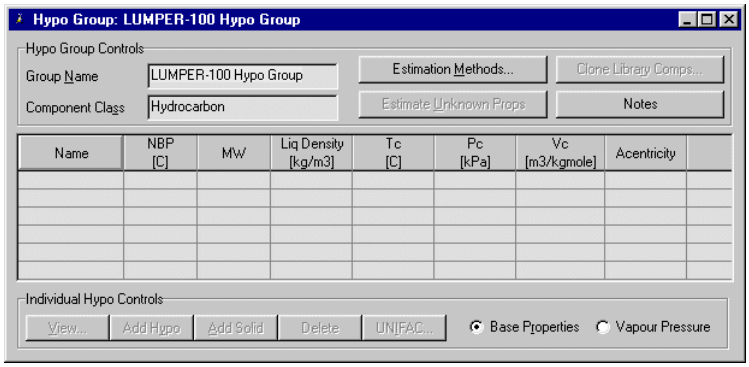
You can create a hypothetical group.

1. Click the **Create Hypo Group** button. The Hypo Group view appears.

The Lumper Hypothetical group name appears on the Hypothetical tab of the Simulation Basis Manager when you enter the Basis environment. Refer to **Section 5.2.3 - Hypotheticals Tab** in the **HYSYS User Guide** for more information.

The Component Class field on the Hypo Group view shows the class for grouping the hypotheticals.

Figure 3.9



The hypothetical group created by the lumper is also managed by the lumper. Therefore, you have very limited access to the functionality of the group.

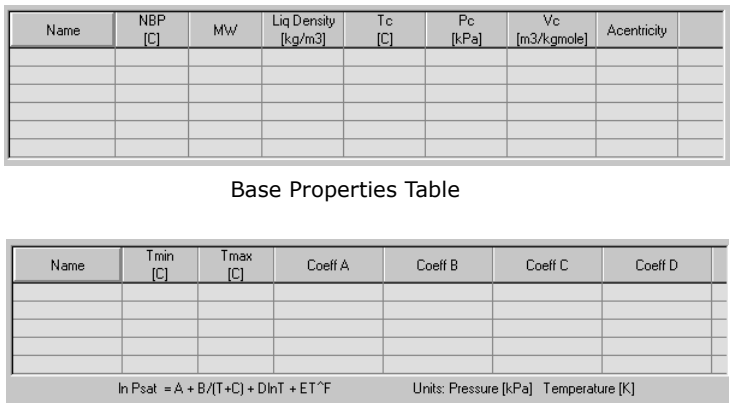
2. If you want to change the Hypo group name, type the new name in the **Group Name** field.

3. In the Individual Hypo Controls group, click the appropriate radio button to select the Basic Hypothetical component properties to view in the table. The table view changes according to the radio button that has been selected.

Refer to the section on the **Base Properties** in the **HYSYS Simulation Basis** guide for more information.

Refer to the section on the **Vapour Pressure Properties** in the **HYSYS Simulation Basis** guide for more information.

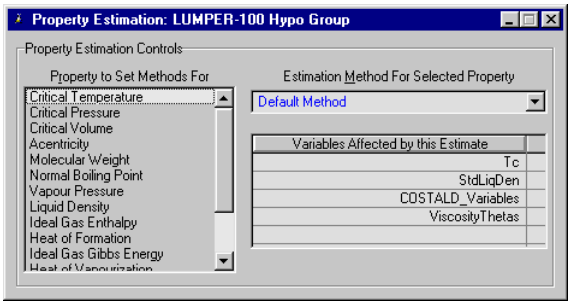
Figure 3.10



4. Click the **Estimation Methods** button to setup property estimation methods for the contained hypothetical components. The Property Estimation view appears.

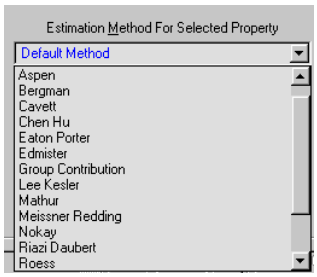
For more information on the Estimation Methods, refer to the section on the **Estimation Methods** in the **HYSYS Simulation Basis** guide.

Figure 3.11



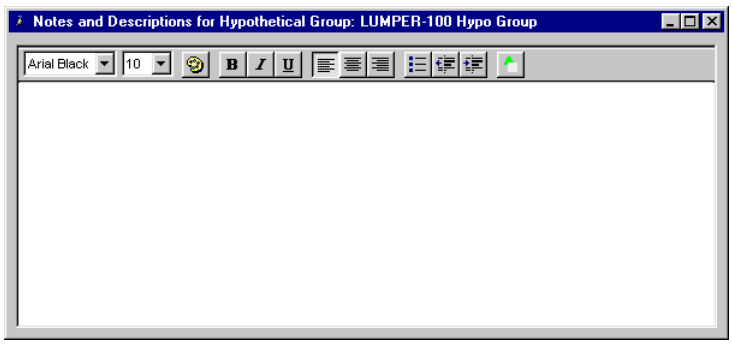
5. On the Property Estimation view, select the property to set the methods for from the Property to Set Methods For list. Then, from the **Estimation Method For Selected Property** drop-down list, select the estimation method. The default estimation method is Default Method.

Figure 3.12



6. Close the Property Estimation view to return to the Hypo Group view.
7. If you want to add notes to the hypothetical group, click the **Notes** button. The Notes and Descriptions for Hypothetical Group view appears.

Figure 3.13



The Notes and Descriptions for Hypothetical Group view provides a text editor that allows you to record any comments or information regarding the hypothetical group. After you have added the notes, close the Notes and Descriptions for Hypothetical Group view to return to the Hypo Group view.

8. Close the Hypo Group view to return to the Lumper property view.

   Once you have closed the Hypo Group view, you will notice that the Create Hypo Group button is now the Edit Hypo Group button and that the Delete Hypo Group and Create User Hypo buttons are now enabled on the Lumper page.

You can also access the hypothetical groups from the Basis environment.

1. From the **Simulation** menu, select **Enter Basis Environment**. The Simulation Basis Manager appears.
2. Click on the **Hypotheticals** tab.
3. Select the hypothetical group you want to edit from the Hypotheticals Groups list.
4. Click the **View** button. The Hypo Group View appears.

Editing a Hypothetical Group

1. Click the **Edit Hypo Group** button. The Hypo Group view appears.
2. From here you can change the estimation methods, add notes, change the group name and the basic hypothetical component properties.

Deleting a Hypothetical Group

1. Click the **Delete Hypo Group** button.
2. HYSYS will prompt you to confirm the deletion of the hypo group.

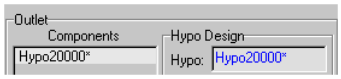
Deleting the hypothetical group will also delete all the hypothetical components it contains, including both user created and auto-generated hypothetical components.

Creating a User Lumped Hypothetical Component

After you have created a hypothetical group, you can create a new lumped hypothetical component.

1. Click the **Create User Hypo** button.
2. The created lumped hypothetical component is displayed in the outlet components list. The Hypo field of the Hypo Design group allows you to change the name of the lumped hypothetical component.

Figure 3.14



Deleting a User Lumped Hypothetical Component

1. From the list of components in the Outlet group, select the lumped hypothetical component you want to delete.
2. Click the **Delete User Hypo** button.

HYSYS does not prompt you to confirm the deletion of your lumped hypothetical component. After the lumped hypothetical component is deleted, the information cannot be retrieved.

Viewing a Lumped Hypothetical Component

You can also view the lumped hypothetical property view from the Basis environment.

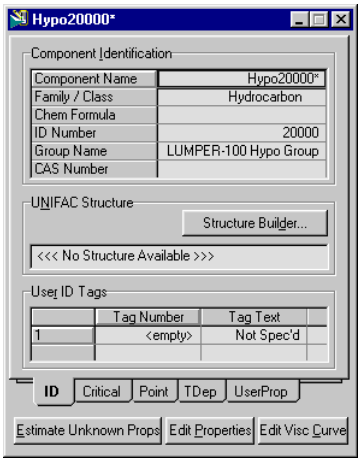
1. From the **Simulation** menu, select **Enter Basis Environment**. The Simulation Basis Manager appears.
2. Click on the **Hypotheticals** tab.
3. Select a lump hypothetical component from the Hypothetical Quick Reference table and click the **View Hypo** button.

You can view both the user created and auto generated lumped hypothetical components from the Hypo Group view.

You are not allowed to alter any data in a hypothetical component created by a lumper.

1. Click the **Edit Hypo Group** button. The Hypo Group view appears.
2. From the **Individual Controls** group, click the **View** button.
3. The information that appears corresponds to the hypothetical group the lumped hypothetical component is associated with.

Figure 3.15

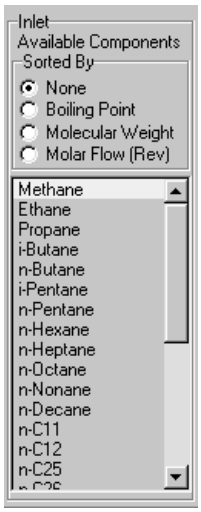


For more information on the lumped hypothetical component property view, refer to **Section 3.5 - Hypothetical Component Property View** in the **HYSYS Simulation Basis** guide.

Adding Components to a User Lumped Hypothetical Component

You can add inlet components to the created lumped hypothetical component.

1. From the list of available components in the Inlet group, select the components you want to add.



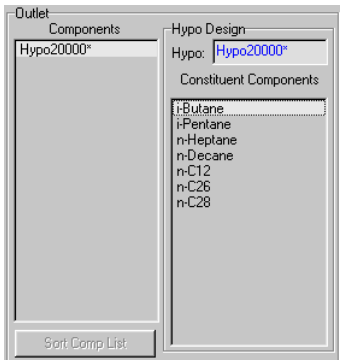
Inlet group

If you want to add all the inlet components to the lumped hypothetical component, click on the first inlet component in the list. Then press SHIFT and click on the last inlet component in the list. All the inlet components will be highlighted. Alternatively, you can also press SHIFT END.

If you want to add individual inlet components to the lumped hypothetical component, press and hold down the CTRL key then click on the inlet components you want to add to the lumped hypothetical component. The selected inlet components will be highlighted.

2. Click the **Add to User Hypo** button.
3. The hypothetical components are displayed in the Constituent Components list of the Hypo Design group.

Figure 3.16



Ensure that you have a lumped hypothetical component selected in the outlet components list before you can remove the component.

Removing a Component from the User Lumped Hypothetical Component

1. From the list of constituent components in the Outlet group, select the components you want to remove.
2. Click the **Remove from Hypo** button. The components will now be moved back to the Available Components list in the Inlet group.

Keeping All Inlet Components

You can add all inlet components to the output component list if you have not created a lumped hypothetical component by clicking the Keep All Comps button.

Keeping Individual Components

You can add individual components from the inlet components list to the output component list.

1. From the available components list of the Inlet group, select the component you want to add to the output component list.
2. Click the **Add to Keep** button. The selected inlet component is added to the outlet component list of the Outlet group.

Ensure that you have a component selected in the outlet components list before you can remove the component.

Removing Components

1. From the components list in the Outlet group, select the individual component you want to remove.
2. Click the **Remove Comp** button. The component will now be displayed in the inlet component list.

Sorting the Outlet Component List

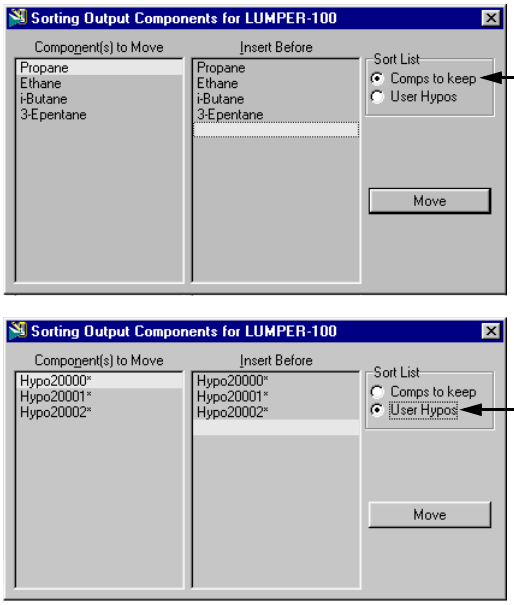
If you have one individual component and one lumped hypothetical component the Sort Comp List button remains disabled.

You cannot sort the automatically lumped hypothetical components.

You can only sort the outlet component list if you have at least two individual components or two user lumped hypothetical components.

1. Click the **Sort Comp List** button in the Outlet group. The Sorting Output Components view appears.
2. From the **Sort List** group, click the appropriate radio button. You can sort the individual components or user lumped hypothetical components in the outlet component list.

Figure 3.17



When you click the Comps to Keep radio button, the Sorting Output Components view shows the individual components that you can sort in the outlet components list.

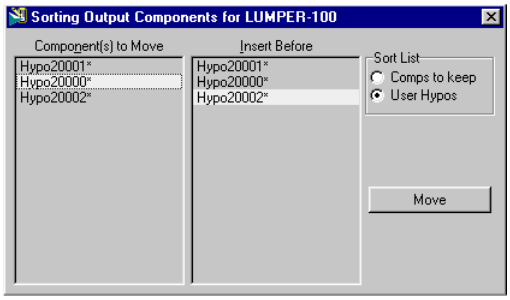
When you click the User Hypos radio button, the Sorting Output Components view shows the user lumped hypothetical components that you can sort in the outlet components list.

3. From the Component(s) to Move list, select the component you want to move. Then, from the Insert Before list, select the component that you want the moved component to be inserted before.
4. Click the **Move** button to move the component.

   From example, if you have three lumped hypothetical components as shown in **Figure 3.17**. When you select Hypo20000 from the Component(s) to Move list, then you select Hypo20002 from the Insert before list and click the

Move button, Hypo20000 will be inserted between Hypo20001 and Hypo20002 as shown in the figure below.

Figure 3.18



5. Close the Sorting Output Components view to return to the Lumper property view.

Emptying and Deleting an Auto Lumped Hypothetical Component

The desired total number of output components is indicated in the Desired total number of components field.

The Empty Auto Hypo and Delete Auto Hypo buttons are active only when you have selected an automatically lumped hypothetical component in the outlet component list.

If the desired total number of outlet components is more than the sum of the individual components and the user lumped hypothetical components, HYSYS will make up the difference by automatically generating the required number of hypos using the chosen lumping method. You can remove all the components from an automatically lumped hypothetical component.

1. From the outlet components list, select the automatically lumped hypothetical component you want to remove all components from.
2. Click the **Empty Auto Hypo** button. The components will now be displayed in the inlet component list.

Ensure that you have an automatically lumped hypothetical component selected in the outlet components list before you can delete.

To delete the automatically lumped hypothetical component:

1. From the outlet components list, select the automatically lumped hypothetical component you want to delete.
2. Click the **Delete Auto Hypo** button.

HYSYS does not prompt you to confirm the deletion of your automatically lumped hypothetical component. After the automatically lumped hypothetical component is deleted, the information cannot be retrieved.

Calculating the Lumper Unit Operation

If you want to test what happens with the current changes, click the Submit Changes button. If you have finished editing, close the view to enable on-the-fly calculations.

The Submit Changes button is enabled when there is sufficient information for the lumper to calculate. By clicking this button, HYSYS will calculate the unit operation based on the current information. Therefore, the Submit Changes button can be viewed as an ad hoc test button.

Closing the Lumper Property View and On-the-Fly Calculations

When the Lumper property view is open, HYSYS assumes that the lumper is still in editing mode and most of the calculations are on hold. Therefore, you must click the Submit Changes button to calculate the unit operation. If you have finished editing and want to enable on-the-fly calculations, you have to close the Lumper property view.

User Variables Page

The User Variables page allows you to create and implement variables in the HYSYS simulation case. For more information on implementing the User Variables, refer to **Chapter 5 - User Variables** in the **HYSYS Customization Guide**.

Notes Page

The Notes page provides a text editor that allows you to record any comments or information regarding the specific unit operation, or the simulation case in general. For more information, refer to **Section 7.20 - Notes** in the **HYSYS User Guide**.

3.1.3 Rating Tab

This unit operation currently does not have rating features.

3.1.4 Worksheet Tab

Refer to **Section 1.3.1 - Worksheet Tab** in the **HYSYS Operations Guide** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

3.1.5 Dynamics Tab

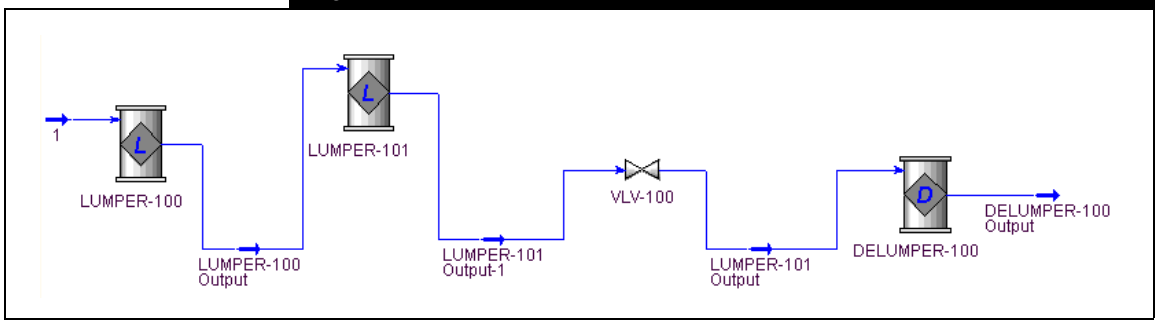
This unit operation is currently not available for dynamic simulation.

3.2 Delumper

For information on the Lumper unit operation, refer to **Section 3.1 - Lumper**.

The Delumper is an upstream unit operation used to delump the lumped hypothetical components created by one or more lumpers.

Figure 3.19



You can attach one or more feed streams using one or more different fluid packages (hence potentially different component lists), with a total of *m* distinct components. In the Delumper property view, you can specify a new equation of state (EOS) fluid package, a new fluid package name and a new components list name.

By using the currently available Component Recovery delumping method, you can recover part or all of the original components from the lumped hypothetical components, with a total of *n* outlet components, where $n \geq m$.

The Delumper does not have to be connected to a Lumper directly in order to delump its lumped components. Furthermore, the Component Recovery delumping method is capable of recovering the very original constituent components from a lumped hypothetical component, regardless how many layers of lumping have been conducted.

As a result, the outlet component list may include the individual components from the input list, and recovered components from the lumped components in the input components list.

3.2.1 Delumper Property View

There are two ways that you can add a delumper to your simulation:

You can also add the delumper by clicking the **F12** hot key.

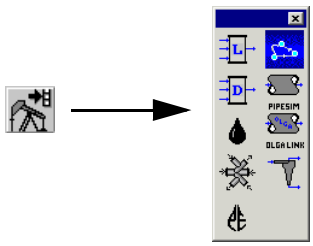
1. From the **Flowsheet** menu, click **Add Operation**. The UnitOps view appears.
2. Click the **Upstream Ops** radio button.
3. From the list of available unit operations, select **Delumper**.
4. Click the **Add** button.

OR

You can also open the Object Palette by clicking the **F4** hot key.

1. From the **Flowsheet** menu, click **Palette**. The Object Palette appears.
2. Click on the **Upstream Ops** icon. The Upstream Object Palette appears.

Figure 3.20

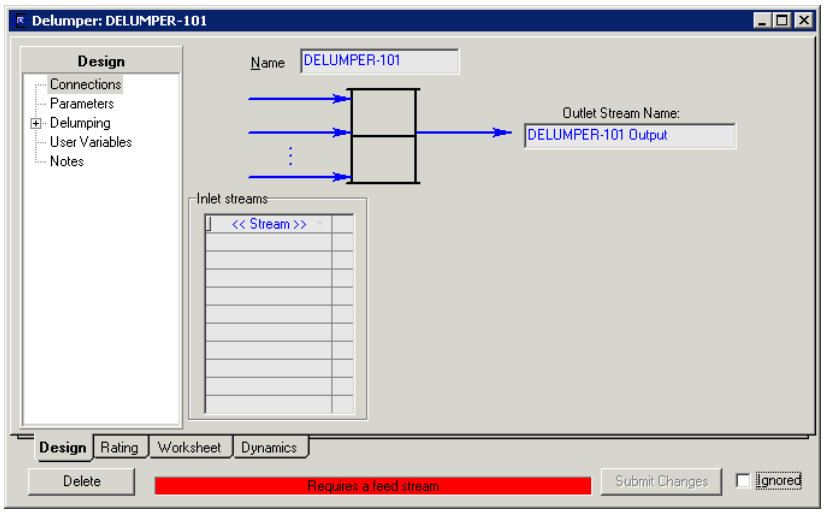


Delumper icon

3. In the Upstream Object Palette, double-click the **Delumper** icon.

The Delumper property view appears.

Figure 3.21



You can also delete a Delumper by clicking on the Delumper icon on the PFD and pressing the DELETE key.

To delete the Delumper operation, click the Delete button. HYSYS will ask you to confirm the deletion.

To ignore the Delumper during calculations, activate the Ignored checkbox. HYSYS completely disregards the operation (and cannot calculate the outlet stream) until you restore it to an active state by deactivating the checkbox.

3.2.2 Design Tab

The Design tab consists of the following pages:

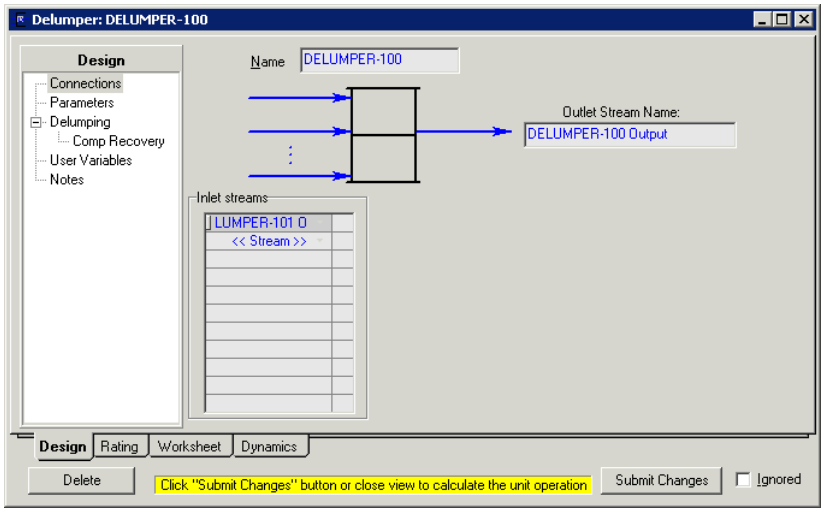
- Connections
- Parameters
- Delumping
- User Variables
- Notes

Connections Page

You can specify one or more inlet streams using one or more different fluid packages. The fluid packages may have different component lists.

The Connections page is used to define all of the connections to the Delumper. You can specify the inlet streams to attach to the operation. The name of the operation can be changed in the Name field, and you can also change the name of the outlet stream in the Outlet Stream Name field.

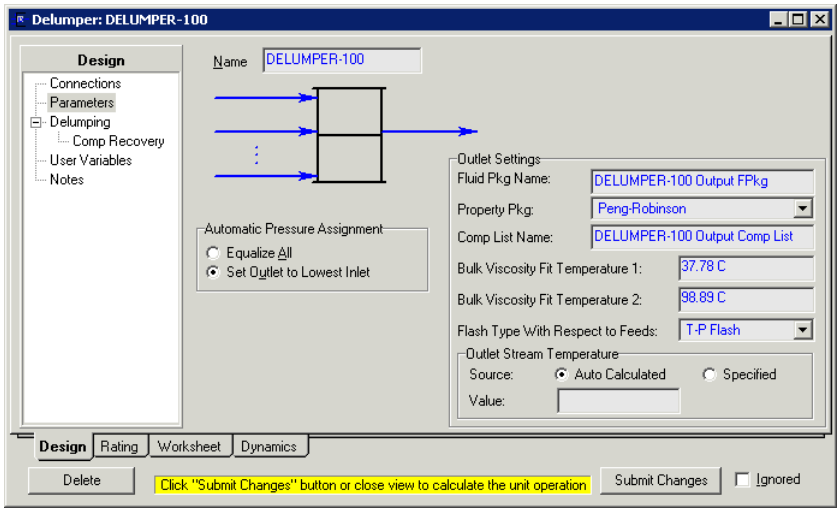
Figure 3.22



Parameters Page

The Parameters page allows you to specify the outlet settings and automatic pressure assignment.

Figure 3.23



The Outlet Settings group consists of the following parameters:

Parameters	Description
Fluid Pkg Name	Allows you to specify the name for the fluid package to be generated.
Property Pkg	You can select the Equation of State (EOS) property package from the drop-down list: • Peng-Robinson • PRSV • Sour PR • SRK • Kabadi-Danner • Sour SRK A new fluid package with components will be created during the delumping process.
Comp List Name	Allows you to specify the name for the component list to be generated.
Bulk Viscosity Fit Temperature 1	The first temperature at which the outlet stream bulk viscosity is to match that of the combined feed.

For more information on the EOS property packages, refer to the section on the **Equations of State (EOS)** in the **HYSYS Simulation Basis** guide.

Parameters	Description
Bulk Viscosity Fit Temperature 2	The second temperature at which the outlet stream bulk viscosity is to match that of the combined feed.
Flash Type with Respect to Feeds	You can select the flash type to be used for the outlet stream with respect to the combined feed. There are two options: • T-P Flash • P-H Flash
Outlet Stream Temperature Group	This group appears when you select T-P as the flash type from the Flash Type with Respect to Feeds drop-down list. There are two radio buttons in the Outlet Stream Temperature group: • **Auto Calculated**. The outlet stream temperature will be automatically calculated based on the combined feed. You cannot specify the outlet stream temperature in the Value field. • **Specified**. You have to specify the outlet stream temperature in the Value field.

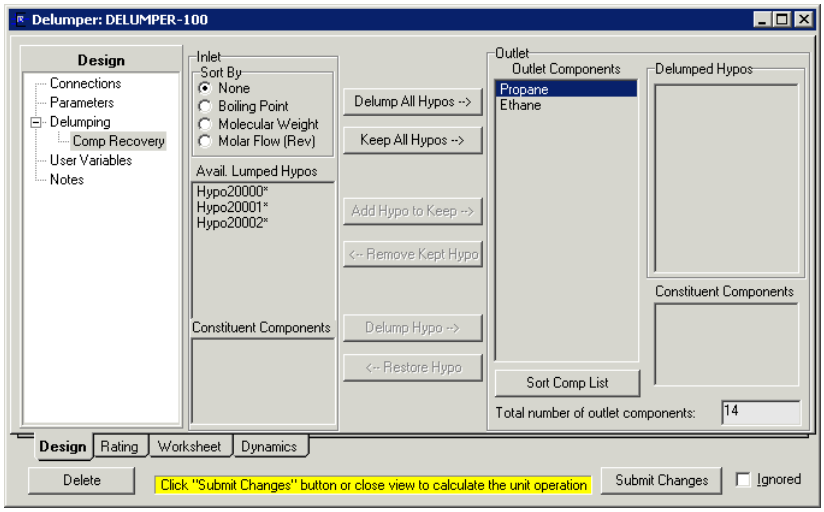
The Automatic Pressure Assignment group consists of the following radio buttons:

- **Equalize All**. Click this radio button, if you want to force all stream pressure to be the same.
- **Set Outlet to Lowest Inlet**. Click this radio button, if you want the outlet pressure to be the lowest inlet pressure.

Delumping Page

On the Delumping page, you can select the delumping method that you want to use.

Figure 3.24



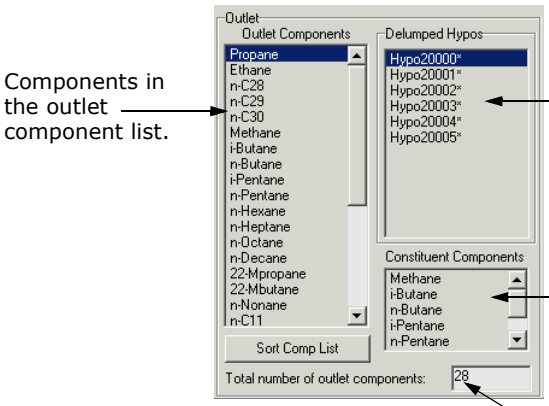
There is currently one delumping method:

For the Component Recovery method only the lumped hypothetical components can be delumped.

Component Recovery: You may choose to recover none to all of the lumped input hypothetical components. The default is to recover all lumped hypothetical components to their original constituent components, regardless how many times the components have been lumped, as well as how many intermediate unit operations exist between the lumpers and the current delumper.

The outlet component list may include the individual components from the input list, and recovered components from the lumped components in the input components list.

Figure 3.25



Sorting the Available Lumped Hypothetical Components

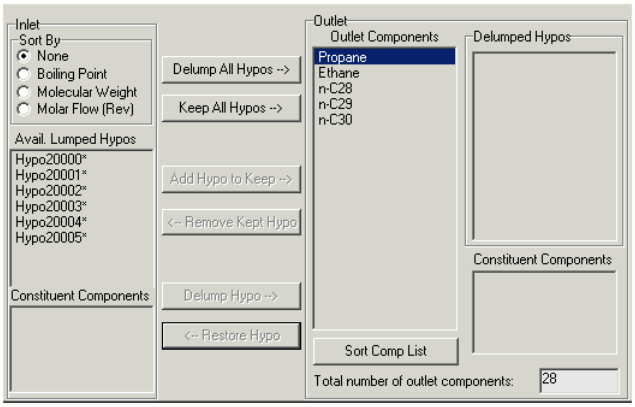
You can sort the available lumped hypothetical components in the inlet group by clicking on the appropriate radio button.

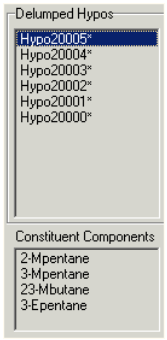
Radio Button	Description
None	The components remain in the original order. This is the order when you added the components to the fluid package component list.
Boiling Point	You can sort the components according to the boiling point of each component.
Molecular Weight	You can sort the components according to the molecular weight of each component.
Molar Flow (Rev)	You can sort the components according to the molar flow of each component. The component with the largest molar flow is on the top.

Delumping All Available Lumped Hypothetical Components

You can delump all the lumped hypotheticals that are shown in the available lumped hypothetical list of the Inlet group.

Figure 3.26





To delump all the available lumped hypothetical components:

1. Click the **Delump All Hypos** button.
2. The delumped hypothetical components appear in the Delumped Hypos group.

Delumping Available Lumped Hypothetical Components

You can delump the lumped hypothetical components from the inlet components list to the delumped hypos list.

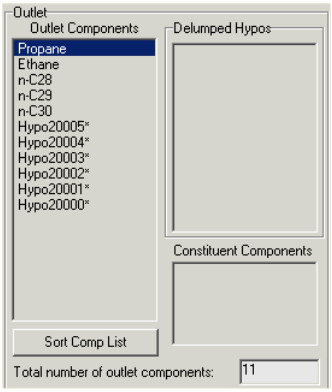
1. From the Avail. Lumped Hypos list of the Inlet group, select the lumped hypothetical component you want to delump.
2. Click the **Delump Hypo** button. The selected lumped hypothetical component appears in the Delumped Hypos group.

Keeping All Available Lumped Hypothetical Components

You can keep all the lumped hypotheticals that are shown in the available lumped hypothetical list of the Inlet group.

1. Click the **Keep All Hypos** button.
2. The lumped hypothetical component appears in the outlet components list of the Outlet group.

Figure 3.27



Keeping Available Lumped Hypothetical Components

You can add lumped hypothetical components from the inlet components list to the output component list.

1. From the Avail. Lumped Hypos list of the Inlet group, select the lumped hypothetical component you want to add to the output component list.
2. Click the **Add Hypo to Keep** button. The selected lumped hypothetical component is added to the outlet component list of the Outlet group.

Removing Kept Lumped Hypothetical Components

Ensure that you have the lumped hypothetical component selected in the outlet components list before you can remove the lumped hypothetical component.

1. From the components list in the Outlet group, select the lumped hypothetical component you want to remove.
2. Click the **Remove Kept Hypo** button. The lumped hypothetical component will now be displayed in the inlet component list.

You cannot remove the individual components from the outlet component list.

Restoring the Delumped Hypothetical Components

You can restore the delumped hypothetical components.

1. From the Delumped Hypos group, select the delumped hypothetical component you want to restore.
2. Click the **Restore Hypo** button. The delumped hypothetical component appears in the Avail. Lumped Hypos list of the Inlet group.

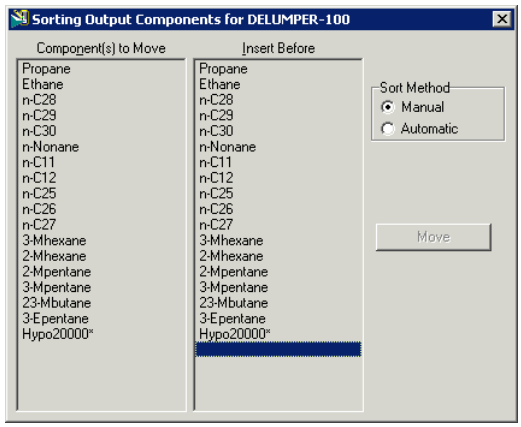
Figure 3.28

Sorting the Output Components

The outlet component list may include the individual components from the input list, and recovered components from the lumped components in the input components list.

You can sort the available components in the Outlet Components list by clicking on the Sort Comp List button. The Sorting Outputs Components for Delumper view appears.

Figure 3.29



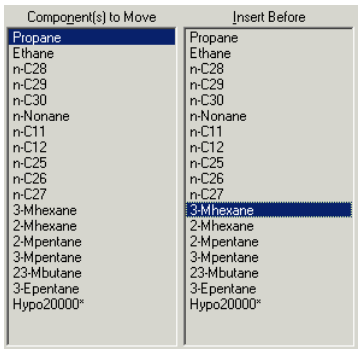
In the Sort Method group, you can select the method you want to use to sort the output components by clicking on the Manual or Automatic radio button.

Manual Sort Method

By default the Manual radio button is selected. To sort manually:

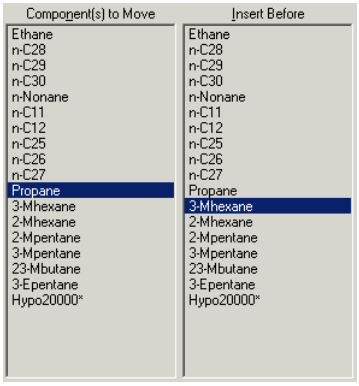
1. From the Component(s) to Move list, select the component you want to move. The **Move** button is enabled.
2. From the Insert Before list, select the component you want the component from the Component(s) to Move list to be inserted before.

Figure 3.30



3. Click the **Move** button. For example, the figure below shows that Propane has been moved before 3-Mhexane.

Figure 3.31

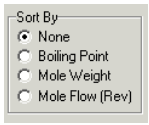


4. Close the Sorting Outputs Components for Delumper view to return to the **Delumping** page.

Automatic Sort Method

When you select the Automatic radio button, the Sort By group appears.

Figure 3.32



Radio Button	Description
None	The components will be put in the order before the sorting view is launched.
Boiling Point	You can sort the components according to the boiling point of each component.
Molecular Weight	You can sort the components according to the molecular weight of each component.
Molar Flow (Rev)	You can sort the components according to the molar flow of each component. The component with the largest molar flow is on the top.

Calculating the Delumper Unit Operation

If you want to test what happens with the current changes, click the Submit Changes button. If you have finished editing, close the view to enable on-the-fly calculations.

The Submit Changes button is enabled when there is sufficient information for the Delumper to calculate. By clicking this button, HYSYS will calculate the unit operation based on the current information. Therefore, the Submit Changes button can be viewed as an ad hoc test button.

Closing the Delumper Property View and On-the-Fly Calculations

When the Delumper property view is open, HYSYS assumes that the delumper is still in editing mode and most of the calculations are on hold. Therefore, you must click the Submit Changes button to calculate the unit operation. If you have finished editing and want to enable on-the-fly calculations, you have to close the Delumper property view.

User Variables Page

The User Variables page allows you to create and implement variables in the HYSYS simulation case. For more information on implementing the User Variables, refer to **Chapter 5 - User Variables** in the **HYSYS Customization Guide**.

Notes Page

The Notes page provides a text editor that allows you to record any comments or information regarding the specific unit operation, or the simulation case in general. For more information, refer to **Section 7.20 - Notes Manager** in the **HYSYS User Guide**.

3.2.3 Rating Tab

This unit operation currently does not have rating features.

3.2.4 Worksheet Tab

Refer to **Section 1.3.1 - Worksheet Tab** in the **HYSYS Operations Guide** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

3.2.5 Dynamics Tab

This unit operation is currently not available for dynamic simulation.

3.3 References

[1] Montel, F and Gouel P.L., *A new Lumping Scheme of Analytical Data for Compositional Studies*. Presented at the 59th Annual Technical Conference and Exhibition, Houston, Sept. 16-19, 1984; Paper SPE 13119.

4 Liquid-liquid Hydrocyclone

4.1 Introduction

The HYSYS Liquid-liquid Hydrocyclone predicts the performance of an oily water cleaning unit operation. The Liquid-liquid Hydrocyclone generates results based on the Migration Probability Theory. An oil droplet size distribution based on a sauter mean diameter is applied and the resulting volume of oil separated is calculated.

The Liquid-liquid Hydrocyclone is designed to be easy to use with a single input tab giving liner details and the oil droplet distribution. Process details, Hydrocyclone liner dimensionless parameters, and separation performance are calculated. You have the option of modelling two different types of liner:

- Vortoil G-liners
- Serck Baker Oilspin liners

The fundamental calculation methods are similar for both liners. The hydraulic parameters however vary considerably.

4.1.1 Theory

The Liquid-liquid Hydrocyclone operation performs the following calculations to generate the results:

- **Oil Droplet Distribution**
- **Hydrocyclone Liner Dimensions**
- **Hydrocyclone Hydraulics**
- **Oil Droplet Migration Probability**
- **Hydrocyclone Separation Efficiency**

Oil Droplet Distribution

The Liquid-liquid Hydrocyclone uses a Rosin Rammler Oil Droplet Distribution to describe the dispersion at the Inlet. A two parameter Cumulative Distribution is defined.

The Cumulative Distribution is defined by the following equation:

$$F(d) = 1 - \exp\left(-\left(\frac{d}{d_{rm}}\right)^{n}\right) \tag{4.1}$$

where:

F(d) = cumulative distribution

d = droplet diameter

d_{rm} *= Rosin Rammler modal diameter*

n = exponential power index

The Rosin Rammler modal diameter d_{rm} can be related to another **mean** diameter d_M by the following equation.

$$d_M = d_{rm} \times [-\ln(1 - f(d))]^{1/n} \tag{4.2}$$

where:

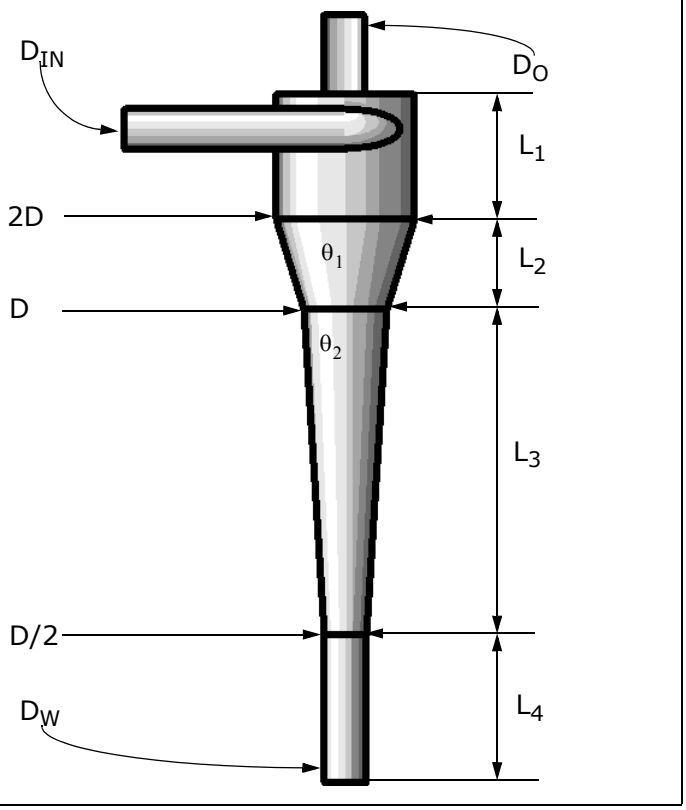
f(d) = fraction undersize at diameter d_M

Hydrocyclone Liner Dimensions

The Hydrocyclone Dimensions are based on the following variables:

- Dimensions Schematic

Figure 4.1



- Characteristic Diameter (D). The Hydrocyclone characteristic diameter is defined by the user.
- Inlet Diameter (D_{IN}). The Inlet diameter is set at 0.35D.
- Underflow Diameter (D_W). The Underflow diameter is set at 0.50D.
- Overflow Diameter (D_O). The Overflow diameter is defined by the user.
- Taper Angles. The Taper angles θ_1 and θ_2 define the Separation section geometry.

- Lengths. The length of each taper section is calculated from the Taper angles (θ_1 and θ_2) and the characteristic diameter (D). The length from the end of the taper section to the liner tip is taken as 20D. These lengths are then summed to give a total liner Length (L).

Hydrocyclone Hydraulics

The Hydrocyclone can be modelled Hydraulically in a dimensionless manner assuming geometrically similar criteria. A Reynolds number and Hydrocyclone number can be defined using dimensions, fluid parameters, and operating conditions. Split Ratio and Maximum flow are also determined from the operating data.

- Reynolds Number. Re_D is expressed as:

$$\mathrm{Re}_D = \frac{Q_T \times \rho_c}{900 \times \pi \times D_H \times \mu_c} \qquad (4.3)$$

where:

Q_T = *volumetric flow rate*

ρ_c = *continuous phase density*

D_H = *Hydrocyclone Characteristic Diameter*

μ_c = *continuous phase viscosity*

- Hydrocyclone Number. Hy_{75} relating to an oil droplet diameter d'_{75} may be defined as:

$$Hy_{75} = \frac{Q_T \times \Delta\rho \times {d'_{75}}^2}{3600 \times {D_H}^3 \times \mu_c} \qquad (4.4)$$

where:

Q_T = *volumetric flow rate*

$\Delta\rho$ = *oil and water density difference*

d'_{75} = *75% Migration Probability Droplet Diameter*

D_H = *Hydrocyclone Characteristic Diameter*

μ_c = *continuous phase viscosity*

The Hydrocyclone Number can also be related to the Reynolds number for similar geometric units by means of the following general equation:

$$Hy_{75} = a(\mathrm{Re}_D)^b \tag{4.5}$$

Experimental or Production performance data can be used to establish the values of *a* and *b*. These constants are Liner specific.

- Split Ratio. Split Ratio is calculated from the user defined Pressure Differential Ratio (PDR) by means of a quadratic expression:

$$F = \alpha(PDR)^2 + \beta(PDR) + \gamma \tag{4.6}$$

where:

α,β,γ = *parameter values established from a curve fit to operating data*

- Maximum Flowrate. Maximum flow rate for the system is related to the pressure differential between the Inlet and Reject streams:

$$Q_{MAX} = n_L \times k(P_{IN} - P_{REJ})^n \tag{4.7}$$

where:

n_L = *number of Liners*

P_{IN} = *Inlet pressure*

P_{REJ} = *Overflow pressure*

k, n = constant values established from hydraulic data

Oil Droplet Migration Probability

The method of Dense Dispersion Hydrocyclones is applied to predict the volume of oil separated from the Inlet stream. A Migration Probability for the droplet distribution is derived from statistical theory and a Reduced Migration Probability.

- Migration Probability. For a given Inlet oil droplet distribution the Migration Probability (MP) of a droplet of diameter d microns is defined as the chance that it will be separated in the oil overhead stream. The MP can be related to the Reduced Migration Probability (RMP) and the Split Ratio (F) by the following expression:

$$MP(d) = RMP(d) \times (1 - F) + F \tag{4.8}$$

- Reduced Migration Probability. An analytical function may be fitted to represent the centre of an envelope of experimental curves for a particular liner. This Reduced Migration Probability (RMP) can be represented generally in terms of a normalised droplet diameter Δ_{75} as:

$$RMP = 1 - \exp(a[\Delta_{75} - b]^c) \tag{4.9}$$

where:

a, b, c = constants determined by experiment

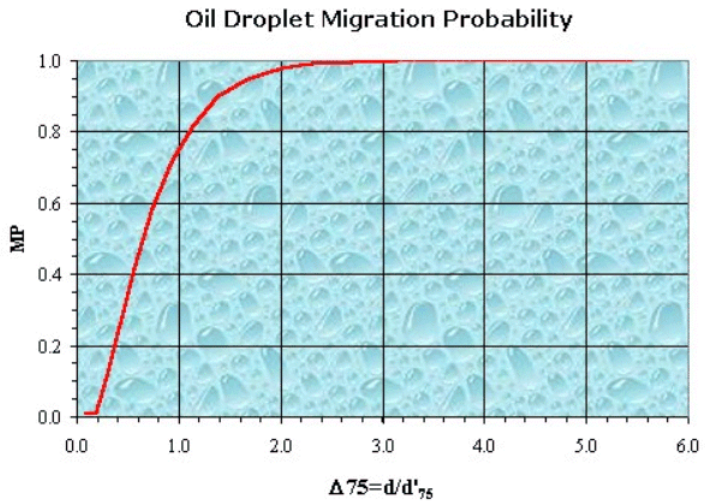
$\Delta_{75} = \frac{d}{d'_{75}}$ *dimensionless droplet diameter*

d'_{75} *= determined from the Hydrocyclone Number*

d = droplet diameter from the Distribution

- Graphical Representation. The Migration Probability for an Oil Droplet Distribution is represented graphically as:

Figure 4.2



Hydrocyclone Separation Efficiency

The Separation Efficiency (ε) of the Hydrocyclone vessel is calculated from Inlet and Outlet streams oil concentrations:

$$\varepsilon = 100\left(1 - \frac{C_o}{C_i}\right) \tag{4.10}$$

where:

C_i = *Inlet stream oil concentrations*

C_o = *Overflow stream oil concentrations*

4.2 Liquid-liquid Hydrocyclone Property View

There are two methods to add a Liquid-liquid Hydrocyclone to your simulation:

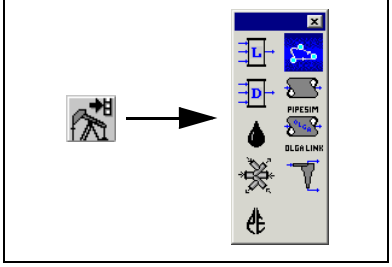
You can also access the UnitOps view by pressing **F12**.

1. From the **Flowsheet** menu, click **Add Operation**. The UnitOps view appears.
2. Click the **Upstream Ops** radio button.
3. From the list of available unit operations, select **Liquid-liquid Hydrocyclone**.
4. Click the **Add** button.

OR

1. From the **Flowsheet** menu, select **Palette** (or press **F4**). The Object Palette appears.
2. In the Object Palette, click the **Upstream Ops** icon to open the Upstream Object Palette.

Figure 4.3

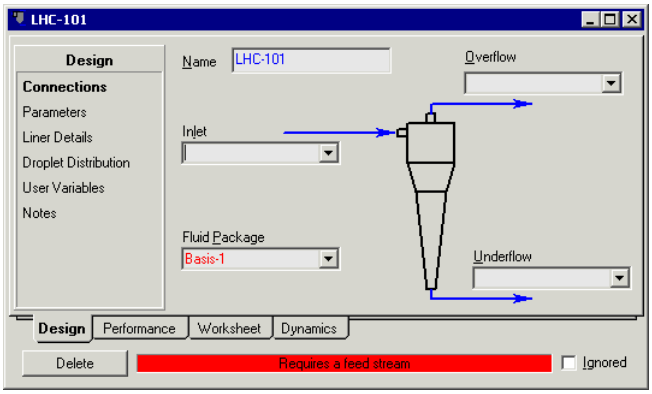


Liquid-Liquid Hydrocyclone icon

3. In the Upstream Object Palette, double-click the **Liquid-Liquid Hydrocyclone** icon.

The Liquid-liquid Hydrocyclone property view appears.

Figure 4.4



You can also delete a Liquid-liquid Hydrocyclone by clicking on the Liquid-liquid Hydrocyclone icon on the PFD and pressing **DELETE**.

To delete the Liquid-liquid Hydrocyclone operation, click the **Delete** button. HYSYS will ask you to confirm the deletion.

To ignore the Liquid-liquid Hydrocyclone during calculations, select the **Ignored** checkbox. HYSYS completely disregards the operation (and cannot calculate the outlet stream) until you restore it to an active state by clearing the checkbox.

4.2.1 Design Tab

The Design tab consists of the following pages:

- Connections
- Parameters
- Liner Details
- Droplet Distribution
- User Variables
- Notes

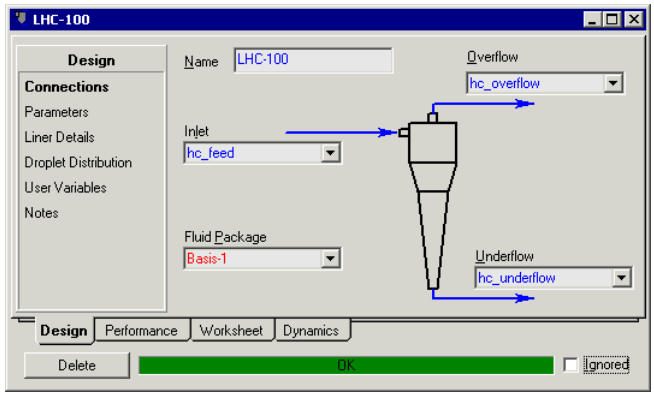
You can select a different fluid package associated to the Liquid-liquid Hydrocyclone using the Fluid Package drop-down list.

Connections Page

The Connections page is used to define all of the connections to the Liquid-liquid Hydrocyclone. You can specify the inlet stream, overflow outlet stream, and underflow outlet stream attached to

the operation. The name of the operation can be changed in the Name field.

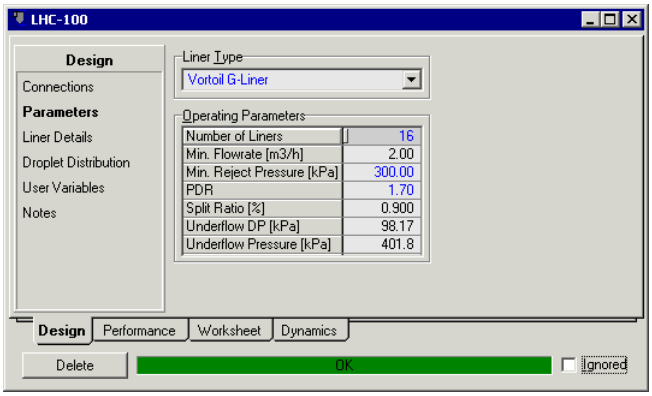
Figure 4.5



Parameters Page

The Parameters page allows you to specify the Liquid-liquid Hydrocyclone operation parameters.

Figure 4.6



The following table lists and describes the parameters available in the Parameters page:

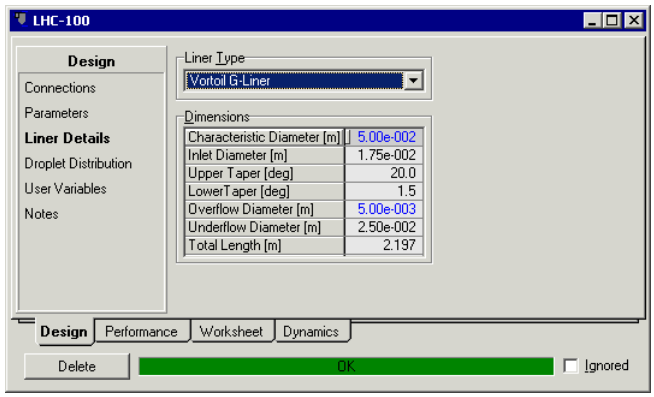
Object	Description
Liner Type drop-down list	Allows you to choose between two types of Vessel liner: • Vortoil G-Liners • Serck Baker Oil Spin Hydraulic parameters and physical Dimensions change between the two types of Liner.
Number of Liners cell	Allows you to specify the number of active vessel liners.
Min. Flowrate cell	Displays the minimum flow rate per liner depending on the selected Liner type. • Vortoil recommends a minimum value of $2m^3/hr$ for the G-Liner. • Serck Baker recommends a minimum value of $4m^3/hr$ for the OilSpin Liner.
Min. Reject Pressure cell	Allows you to specify the minimum Oil Overflow (Reject) downstream pressure.
PDR cell	Allows you to specify the Pressure Differential Ratio. The PDR is the ratio of the following stream pressure drops: $PDR = \frac{\text{Inlet Pressure} - \text{Overflow Pressure}}{\text{Inlet Pressure} - \text{Underflow Pressure}}$

Object	Description
Split Ratio cell	Allows you to specify the volume percent of the total inlet stream that passes to the overflow stream.
Underflow DP cell	Allows you to specify the pressure difference between the inlet stream and the underflow stream.
Underflow Pressure cell	Displays the pressure of the underflow stream.

Liner Details Page

The Liner Details page allows you to manipulate the selected Liner type.

Figure 4.7



The following table lists and describes the parameters available for modification in the Liner Details page:

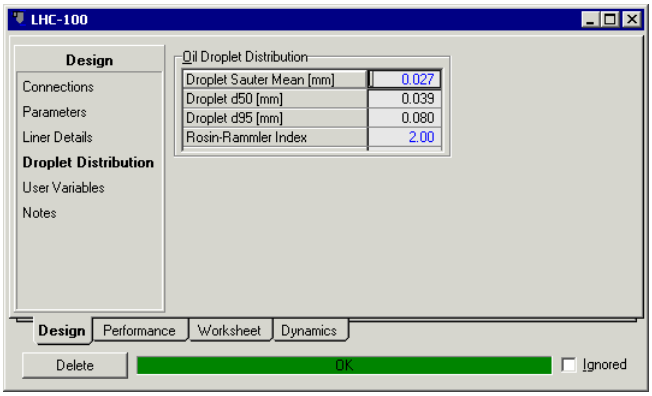
Object	Description
Liner Type drop-down list	Allows you to choose between two types of Vessel liner: • Vortoil G-Liners • Serck Baker Oil Spin Hydraulic parameters and physical Dimensions change between the two types of Liner.
Characteristic Diameter cell	Allows you to specify the liner characteristic diameter, which is used to determine the diameter for the Inlet and Underflow.
Inlet Diameter cell	Displays the calculated inlet diameter value.

Object	Description
Upper Taper cell	Displays the upper taper angle.
Lower Taper cell	Displays the lower taper angle.
Overflow Diameter cell	Allows you to specify the Overflow diameter.
Underflow Diameter cell	Displays the calculated Underflow diameter based on the selected Liner type and the specified characteristic diameter.
Total Length cell	Displays the Liner overall length of the selected Liner type's hydrocyclone geometry.

Droplet Distribution Page

The Droplet Distribution page allows you to manipulate the Liquid-liquid Hydrocyclone performance, by modifying the dispersed oil droplet distribution.

Figure 4.8



The size distribution of oil droplets at the Hydrocyclone inlet is calculated using two parameters of the Rosin Rammler distribution. The Rosin Rammler distribution calculation is based on a mean droplet diameter and an exponential term power index.

The following table lists and describes the distribution parameters:

Parameter	Description
Droplet Sauter Mean	This is the droplet diameter whose volume to surface area ratio is the same as that of the distribution as a whole and so represents the surface area mean diameter.
Droplet d50	This is the diameter of droplet at the 50% undersize point on a cumulative volume distribution curve.
Droplet d95	This is the diameter of droplet at the 95% undersize point on a cumulative volume distribution curve.
Rosin Rammler Index	This is the power term to which the exponential part of the Rosin-Rammler Distribution is raised. Usually the value is between 1 and 2.5.

User Variables Page

The User Variables page allows you to create and implement variables in the HYSYS simulation case. For more information on implementing the User Variables, refer to **Chapter 5 - User Variables** in the **HYSYS Customization Guide**.

Notes Page

For more information, refer to **Section 7.20 - Notes Manager** in the **HYSYS User Guide**.

The Notes page provides a text editor that allows you to record any comments or information regarding the specific unit operation, or the simulation case in general.

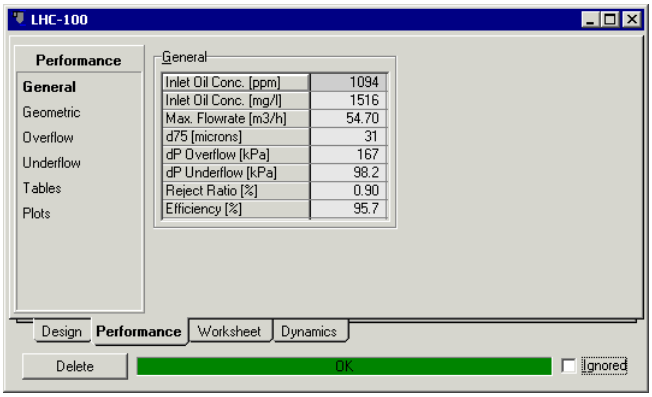
4.2.2 Performance Tab

The Performance tab displays the calculated performance results of the Liquid-liquid Hydrocyclone.

General Page

The General page displays the calculated general Liner performance results.

Figure 4.9

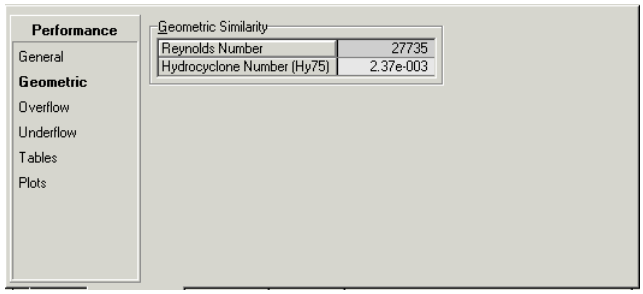


- Inlet oil concentration in parts per million (ppm) by volume and mg/l
- Maximum flow rate for the vessel. This value is calculated from the Liner hydraulic characteristics.
- Droplet diameter separated with 75% efficiency at operating conditions
- Pressure drops at Overflow and Underflow relative to the Inlet
- System Reject Ratio
- System separation efficiency

Geometric Page

The Geometric page displays the calculated geometric Liner performance results.

Figure 4.10

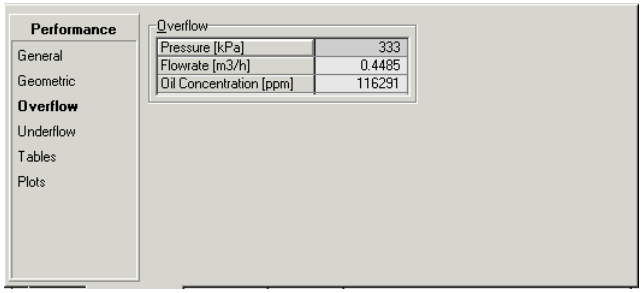


- Hydrocyclone Reynolds Number based on the Characteristic diameter
- Hydrocyclone Number (Hy75)

Overflow Page

The Overflow page displays the calculated Overflow results.

Figure 4.11

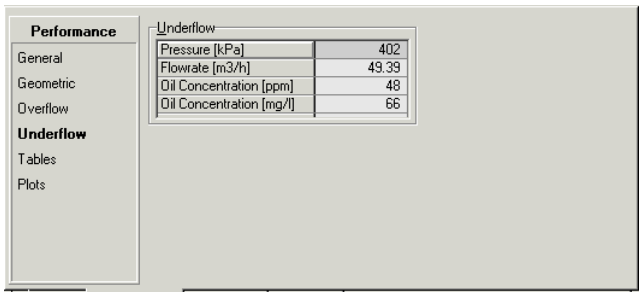


- Overflow pressure
- Volumetric flowrate
- Oil concentration in ppm

Underflow Page

The Underflow page displays the calculated Underflow results.

Figure 4.12

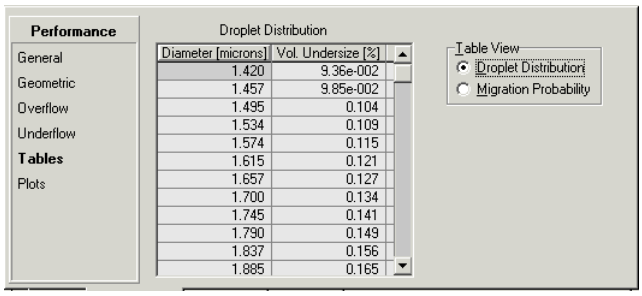


- Underflow pressure
- Volumetric flowrate
- Oil concentration in ppm and mg/l

Tables Page

The Tables page displays the tabulated results of the Oil Droplet Distribution or the Migration Probability. To view either results select the appropriate radio button.

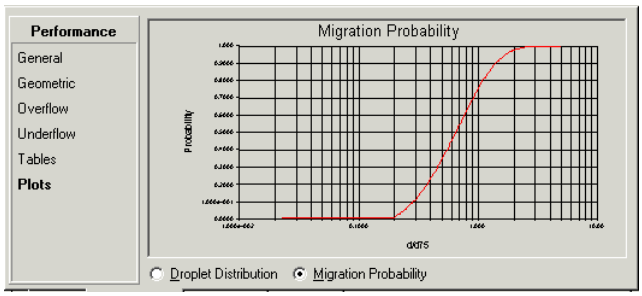
Figure 4.13



Plots Page

The Plots page displays in graph format the results of the Oil Droplet Distribution or the Migration Probability. To view either plot select the appropriate radio button.

Figure 4.14



4.2.3 Worksheet Tab

Refer to **Section 1.3.1 - Worksheet Tab** in the **HYSYS Operations Guide** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

4.2.4 Dynamics Tab

This unit operation is currently not available for dynamic simulation.

4.3 Nomenclature

The following Nomenclature has been adopted for the Liquid-liquid Hydrocyclone calculations:

Variable	Symbol	Units
Volumetric Flowrate	Q_T	m^3/hr
Maximum Volumetric Flowrate	Q_{MAX}	m^3/hr
Inlet Pressure	P_{IN}	Bar
Overflow Pressure	P_{REJ}	Bar
Underflow Pressure	P_{OUT}	Bar
Continuous Phase Density	ρ_c	kg/m^3
Oil Droplet Density	ρ_o	kg/m^3
Hydrocyclone Characteristic Diameter	D	m
Continuous Phase Viscosity	μ_c	Pa.s
Droplet Diameter	d	microns
Sauter Mean Droplet Diameter	$d_{3,2}$	microns
50% Droplet Diameter	d_{50}	microns
75% Droplet Diameter	d_{75}	microns
95% Droplet Diameter	d_{95}	microns
75% Migration Probability Droplet Diameter	d'_{75}	microns
Dimensionless Droplet Diameter	Δ_{75}	
Pressure Differential	ΔP	Bar
Separation Efficiency	ε	%
Inlet Oil Concentration	C_i	ppm Vol.
Underflow Oil Concentration	C_o	ppm Vol.
Split Ratio	F	
Hydrocyclone Reynolds Number	Re_D	
Hydrocyclone Number	Hy_{75}	

Variable	Symbol	Units
Number of Liners	n_L	
Total Liner Length	L	m
Upper Taper Angle	θ_1	degrees
Lower Taper Angle	θ_2	degrees

5 PIPESIM Link

5.1 Introduction

We are pleased to introduce the PIPESIM Single Branch Extension to HYSYS - the first and only commercial coupling between petroleum and process simulation. This extension is a result of a collaborative development between Hyprotech and Baker Jardine. Together we are working on a vision of "Total Hydrocarbon Asset Management". Examining the sensitivity of production and process systems simultaneously or performing production system optimization can yield significant production benefits, sometimes as high as 15% increases in production. We at Hyprotech and Baker Jardine aim to bring you the commercial software tools to achieve these gains.

Hyprotech and Baker Jardine have announced an alliance aimed at producing the first fully integrated, commercial Production System Model. In this, the first product from our alliance, Baker Jardine's industry leading wellbore, tubing, and flowline simulation software, PIPESIM, has been coupled into Hyprotech's HYSYS simulation environment, allowing for the development of models from well description through the entire process production facilities sharing common PVT descriptions from the HYSYS thermodynamics.

Existing PIPESIM production models of wells, flowlines, and risers can be imported into the HYSYS environment and used seamlessly as a HYSYS unit operation using the power of the PIPESIM engine in the background. This first release is limited to a serial string of devices within the PIPESIM model where there is no change in composition; all composition changes are handled within the HYSYS environment. However, one can add multiple PIPESIM extensions to any HYSYS simulation.

5.1.1 How This Chapter Is Organized

This chapter is a comprehensive guide that details all the procedures you need to work with the PIPESIM Link extension. To help you learn how to use PIPESIM Link efficiently, this chapter thoroughly describes the views and capabilities of the PIPESIM Link as well as outlining the procedural steps needed for running the extension. The basics of building a simple PIPESIM Link model is explored in the tutorial (example) problem. The case is presented as a logical sequence of steps that outline the basic procedures needed to build a PIPESIM Link case.

This chapter also outlines the relevant parameters for defining the entire extension and its environment. Each view is defined on a page-by-page basis to give you a complete understanding of the data requirements for the components and the capabilities of the extension.

The PIPESIM Link chapter does not detail HYSYS procedures and assumes that you are familiar with the HYSYS environment and conventions. If you require more information on working with HYSYS, please refer to the **HYSYS User Guide**. In this chapter, you will find all the information you require to set up a case and work efficiently within the simulation environment. If you require more information regarding PIPESIM 2000 please refer to the PIPESIM 2000 reference manuals.

5.1.2 Disclaimer

PIPESIM Link is the proprietary software developed jointly by Hyprotech, a subsidiary of Aspen Technology Inc., (hereafter known as Hyprotech) and Baker Jardine & Associates Limited (hereafter known as Baker Jardine).

Neither Hyprotech nor Baker Jardine make any representations or warranties of any kind whatsoever with respect to the contents hereof and specifically disclaims without limitation any and all implied warranties of merchantability of fitness for any particular purpose. Neither Hyprotech nor Baker Jardine will have any liability for any errors contained herein or for any losses or damages, whether direct, indirect or consequential, arising from the use of the software or resulting from the results obtained through the use of the software or any disks, documentation or other means of utilisation supplied by Hyprotech or Baker Jardine.

Hyprotech and Baker Jardine reserve the right to revise this publication at any time to make changes in the content hereof without notification to any person of any such revision or changes.

5.2 PIPESIM Link Extension

The PIPESIM Link Extension is a unit operation for using the PIPESIM software package used to simulate pipeline systems within the HYSYS framework. The PIPESIM Link functions in the same manner as any HYSYS unit operation or application in terms of its layout and data entry methods. The view consists of three worksheet tabs. At the bottom of each worksheet is a status bar which guides data entry and indicates required information, as well as indicating the status of the PIPESIM simulation once the calculation has been initialized.

PIPESIM has a comprehensive suite of methods and correlations for modeling single and multi-phase flow in production equipment and is capable of accurately simulating a wide range of conditions and situations. You have the option of using the default correlations for the PIPESIM calculations, or specifying your own set from the list of available methods for each parameter. Any change to the PIPESIM models must be done from within the PIPESIM environment.

PIPESIM is fully compatible with all of the gas, liquid, and gas/liquid fluid packages in HYSYS. You can combine PIPESIM and HYSYS objects in any configuration during the construction of a HYSYS flowsheet. PIPESIM objects can be inserted at any point in the flowsheet where single or multi-phase pipe flow effects must be accounted for in the process simulation.

5.2.1 PIPESIM Link Features

The PIPESIM Link extension is functionally equivalent to a HYSYS flowsheet operation. It is installed in a flowsheet and connected to material streams. Unlike a standard pipe segment, an energy stream is not supported. All PIPESIM Link Extension properties are accessed and changed through a set of property views that are simple and convenient to use. The starting point for the definition of a PIPESIM Link Operation, is the PIPESIM Link property view.

The PIPESIM Link property view is where the inlet and outlet material streams are specified. The Inlet Object and Outlet Object fields are read only. These fields are blank when the extension is first installed. Once the PIPESIM Link model has been selected, via the Model page, the names of the boundary objects within the PIPESIM Link model will be displayed

Solution is possible with connection of either a single or both ends of the unit operation. The following specifications are supported in addition to the inlet temperature that must always be specified if a connection.

- **Inlet Connection**: Inlet flow or inlet pressure
- **Outlet Connection**: Outlet flow or outlet pressure (Outlet flow specification is only supported for models in which as Adder/Multiplier operation is not used)
- **Both Ends**: Inlet flow and inlet pressure
- Inlet flow and outlet pressure
- Inlet pressure and outlet pressure

The HYSYS specifications will always override any specifications made within the PIPESIM model.

The following restrictions may surprise experienced HYSYS users:

- An outlet flow specification is not the same as an inlet flow specification since the PIPESIM Link model may manipulate the flow such that the inlet does not equal the outlet flow.
- Negative flowrate cases are not supported for modeling reverse flow.

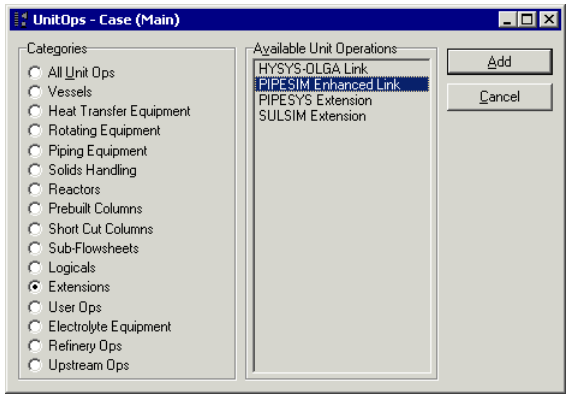
5.2.2 Adding the PIPESIM Link

For further details on creating a HYSYS case, refer to the Basis Environment chapter in the **HYSYS User Guide**.

To add a PIPESIM Link Extension to a HYSYS case:

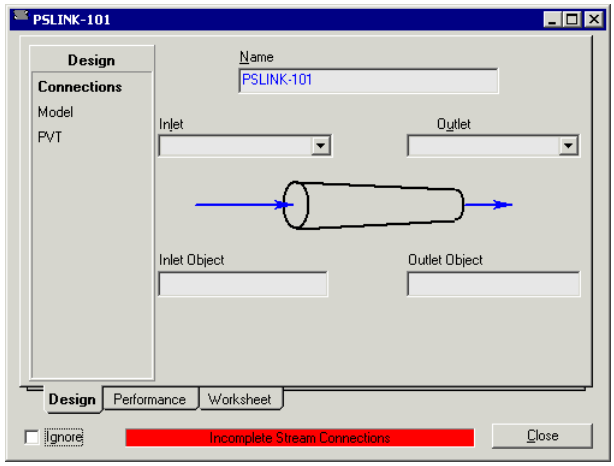
1. Create a HYSYS case suitable for the addition of the PIPESIM Link Extension. As a minimum, you must create a case with a fluid package and two material streams.
2. From the **Flowsheet** menu, select **Add Operation**. The UnitOps view appears.
3. From the **Categories** group, select the **Extensions** radio button.
4. From the **Available Unit Operations** group, select **PIPESIM Enhanced Link**.

Figure 5.1



5. Click the **Add** button. The PIPESIM Link property view appears.

Figure 5.2



6. On the **Connections** page of the **Design** tab, select the material streams from the **Inlet** and **Outlet** drop-down lists. If you have not yet installed these streams in the case, type the material stream names in the **Inlet** and **Outlet** fields.
7. To define the stream conditions, click on the **Worksheet** tab and then the **Conditions** page.

Following these steps allows you to complete the installation of a PIPESIM Link Extension. Once the calculations are complete the Object Status bar will be green and state OK.

5.2.3 PIPESIM Link User Interface

The PIPESIM Link user interface is completely integrated into the HYSYS environment and conforms to all HYSYS usage conventions for operations and data entry. If you are an experienced user of HYSYS, you will already be familiar with all of the features of the PIPESIM Link user interface. If you are a new user, you should begin by studying the **HYSYS User Guide**, since you will need to learn more about HYSYS before you can use the PIPESIM Link Extension.

Like all HYSYS property views, the PIPESIM Link property view allows you access to all of the information associated with a particular item. The view has a number of tabs and on each tab are pages of related parameters.

5.2.4 PIPESIM Link Property View

The PIPESIM Link property view allows you to enter the data that defines the basic characteristics of a PIPESIM Link operation.

This setting is recommended if you have a slow computer and data processing is slowing down the entry process or if you want to delay the calculations until you have entered all of your data.

The the Ignore checkbox at the bottom of the property view can be checked if you want to disable the concurrent calculation of intermediate results during data entry. HYSYS completely disregards the operation until you restore it to an active state by deactivating the checkbox.

The PIPESIM Link property view is the starting point for the definition of any PIPESIM Link operation. The PIPESIM Link property view consists of the following tabs:

- Design
- Performance
- Worksheet

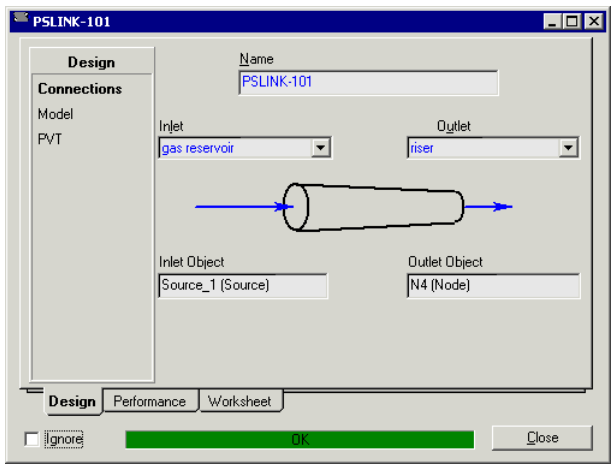
Design Tab

The Design tab is used to define the connections between the HYSYS simulation case and the PIPESIM Link operation, to import and export PIPESIM cases, and to view the basis for tabular physical properties.

Connections Page

The Connections page allows you to select the input and output material streams using the drop-down list or by typing the new material streams in the Inlet and Outlet fields. You can also enter a name for the operation in the Name field.

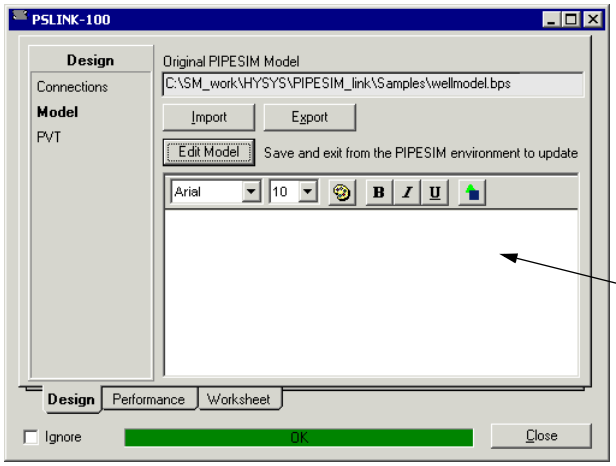
Figure 5.3



Model Page

The Model page allows you to import, export, and edit a PIPESIM model.

Figure 5.4



The Original PIPESIM Model field is read only and echoes the original path and file name of the model that is imported via the Import button.

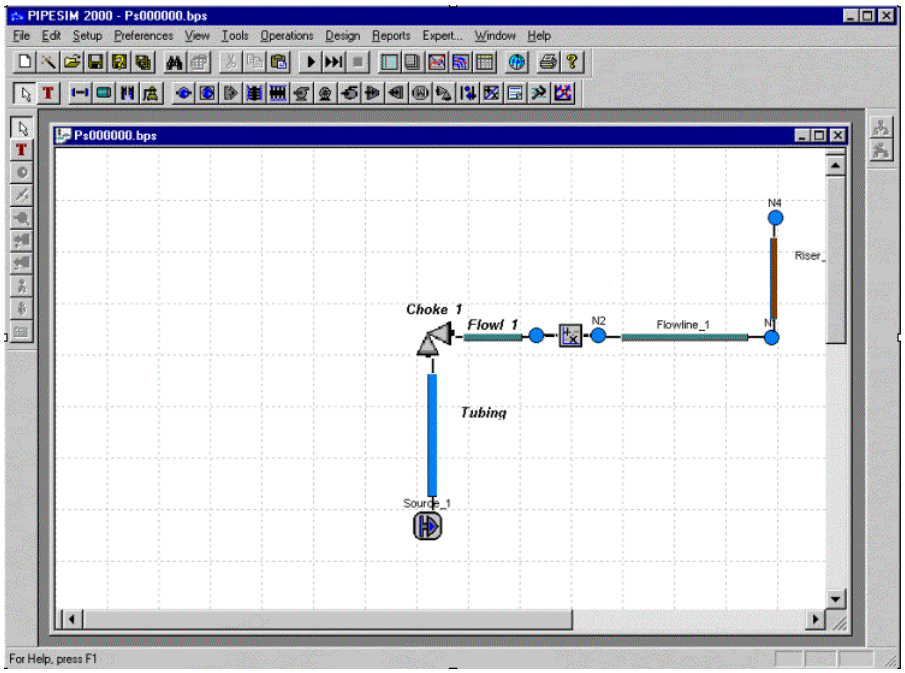
The Original PIPESIM Model field is for documentation purposes only since the model is embedded within the HYSYS model rather than referenced at the location in this field. It may be empty if the model has been created from within the link, rather than by importing an existing PIPESIM model.

The Import button allows you to import an existing PIPESIM model. Clicking the Import button opens the standard Window file picker view that allows model selection. Only single branch models identified by the *.bps extension can be selected.

The Edit Model button loads and runs the PIPESIM2000 GUI with the current model. If a model has not been imported then the PIPESIM2000 GUI contains an empty model. When the GUI is loaded, a work file name appears in the caption bar. The model

can be freely edited, but in order for changes to be reflected in the HYSYS flowsheet, it must be saved under the same file name.

Figure 5.5



The PIPESIM2000 GUI appears modally, so HYSYS does not continue its calculation until the GUI is closed. The PIPESIM model is saved with the HYSYS case and not recorded stand alone.

The Export button allows you to make a copy of the PIPESIM model reflecting any changes made since importing it. Clicking the Export button opens the standard Windows file picker view that allows selection of the file name. Only single branch models identified by the *.bps extension can be selected. Notice that only the configuration of the pipes and nodes will be changed. The source composition data is not changed to reflect the current composition of the feed stream in the HYSYS flowsheet.

The PIPESIM model may not contain any unit operations that change the compositions of the fluid because the PVT table corresponds to a single composition. Thus the following restrictions apply:

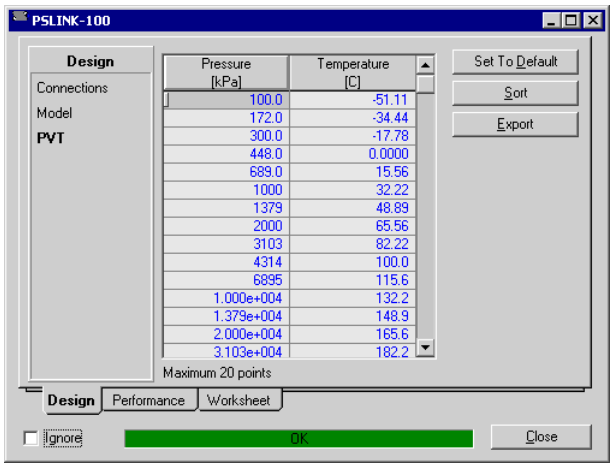
- Vapour/Liquid separators are not supported.
- Well completions cannot use gas lift.

If any of these constraints are violated the status bar gives an appropriate indication and the link will not attempt to solve.

PVT Page

The PIPESIM model solves using tabular physical properties generated by the chosen HYSYS property package. The PVT table defines the extents and granularity of the table. The table is regenerated using the current feed composition each time the link resolves. A maximum of 20 pressure and 20 temperature points can be specified.

Figure 5.6



If you want to set the values of the pressure and temperature points to correspond to the internal PIPESIM default values, click the Set to Default button. The pressure and temperature default values are::

Pressure (psia)									
14.50	24.95	43.51	64.98	99.93	145.0	200.0	290.1	450.1	625.7
1000	1450	2000	2901	4500	6527	10000	1450	20000	29010

Temperature ($^{\circ}$F)									
-60	-30	0	32	60	90	120	150	180	212
240	270	300	330	360	390	420	450	480	510

Calculations do not start unless the data is in ascending order. If the data is not sorted in ascending order, you can sort the data by clicking on the Sort button.

The Export button allows you to export the PVT table file for standalone use with PIPESIM.

Performance Tab

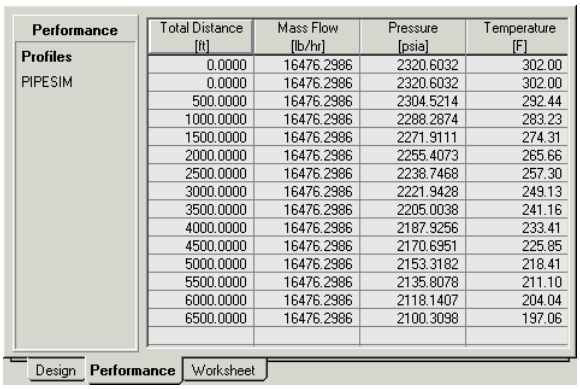
The Performance tab contains two pages that allow you to view a table of the profile data and to view the traditional PIPESIM output.

Profiles Page

The Profiles page displays a tabular view of the profile data for the following key variables vs. axial distance along the pipe:

- Mass Flow
- Pressure
- Temperature
- Holdup (actual volumetric)
- Velocity

Figure 5.7



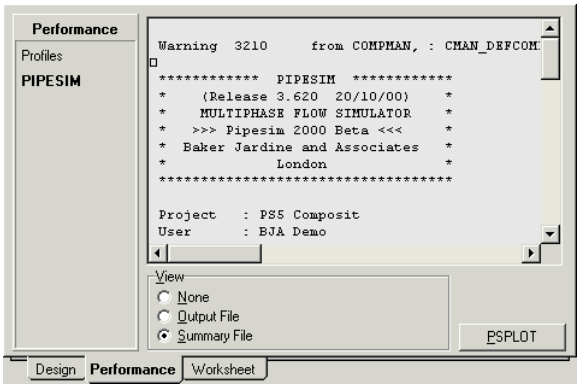
Total Distance [ft]	Mass Flow [lb/hr]	Pressure [psia]	Temperature [F]
0.0000	16476.2986	2320.6032	302.00
0.0000	16476.2986	2320.6032	302.00
500.0000	16476.2986	2304.5214	292.44
1000.0000	16476.2986	2288.2874	283.23
1500.0000	16476.2986	2271.9111	274.31
2000.0000	16476.2986	2255.4073	265.66
2500.0000	16476.2986	2238.7468	257.30
3000.0000	16476.2986	2221.9428	249.13
3500.0000	16476.2986	2205.0038	241.16
4000.0000	16476.2986	2187.9256	233.41
4500.0000	16476.2986	2170.6951	225.85
5000.0000	16476.2986	2153.3182	218.41
5500.0000	16476.2986	2135.8078	211.10
6000.0000	16476.2986	2118.1407	204.04
6500.0000	16476.2986	2100.3098	197.06

PIPESIM Page

The PIPESIM page displays the traditional PIPESIM output. The type of view shown depends on the radio button you have selected in the View group:

- **None**. Select the None radio button, if the link is within a recycle and/or adjust loop to minimise the overhead of importing the results files into the display widget.
- **Output File**. Displays all results and an echo of the PIPESIM model data.
- **Summary File**. Displays a summary of the calculated pressure/temperature/flow profile.

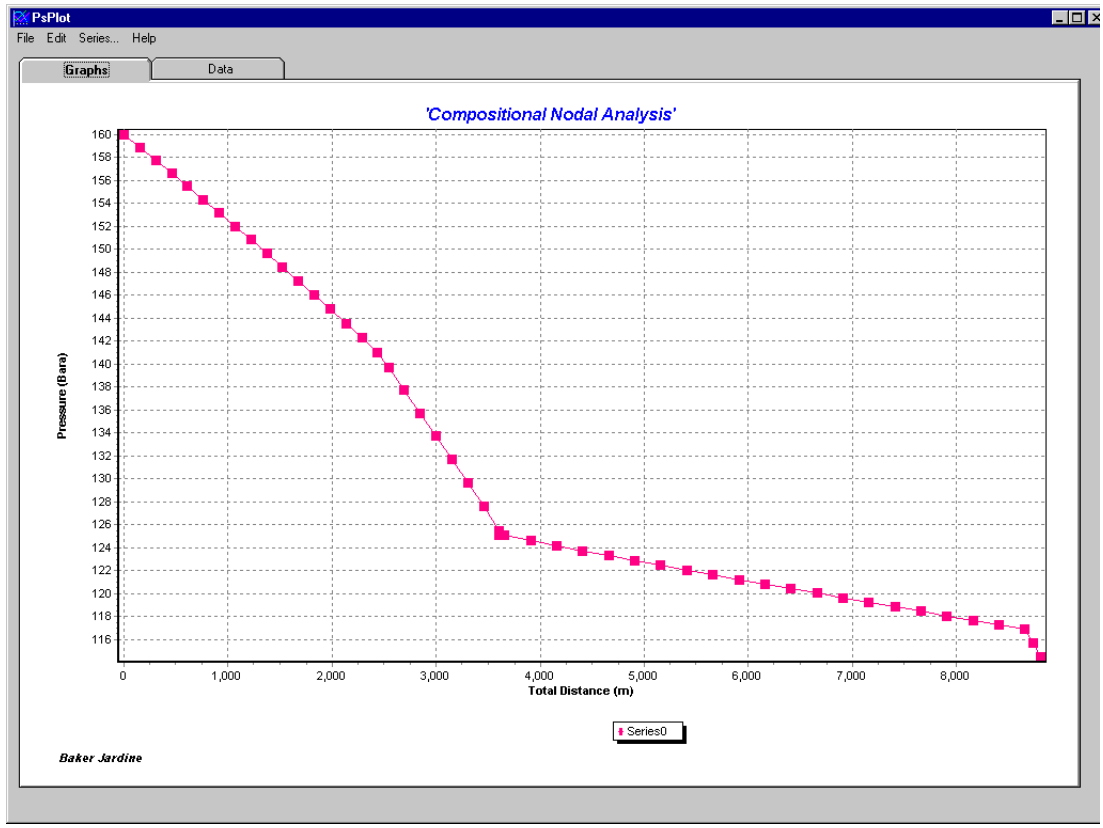
Figure 5.8



Pipe profiles can also be viewed by clicking the PSPLOT button on the PIPESIM page. The PSPLOT button loads and runs the PIPESIM plotting utility.

The PIPESIM plotting utility can be used to display any of the profile results that have been calculated by PIPESIM.

Figure 5.9



The PSPLOT appears modally, so HYSYS does not continue its calculations until the PSPLOT is closed.

Worksheet Tab

Refer to the **Section 1.3.1 - Worksheet Tab** in the **HYSYS Operations Guide** for more information on the Worksheet tab.

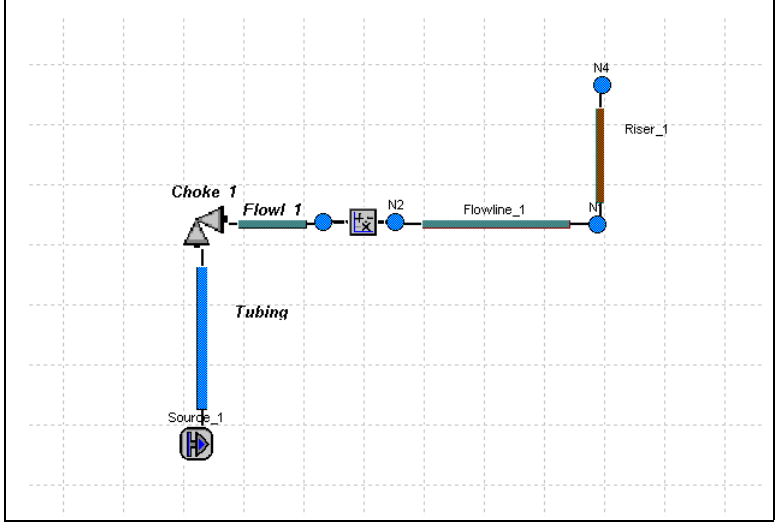
The Worksheet tab allows you to directly edit the material streams that are attached to the PIPESIM Link operation without having to open the material streams property views.

5.3 PIPESIM Link Tutorial

The purpose of the tutorial is to insert a PIPESIM pipeline into HYSYS that has a series of connecting components. In this example, you will go through the steps of importing a PIPESIM model into HYSYS. All units of measurement in this example are SI, but you can change these to whatever unit system you are accustomed to using.

For this case, a simple PIPESIM pipeline consisting of a fluid source, a tubing, a choke, a flowline, an operator, a second flowline, and a riser will be imported into HYSYS. The figure below shows the PIPESIM piping schematic of the system.

Figure 5.10



5.3.1 Flowsheet SetUp

Before working with the PIPESIM Link Extension, you must first create a HYSYS case.

1. In the Simulation Basis Manager, create a fluid package using the Peng Robinson equation of state (EOS). Add the components methane, ethane, propane, i-butane, n-butane, i-pentane, n-pentane, n-hexane, nitrogen, carbon dioxide, and hydrogen sulphide.

Property Package	Components
Peng Robinson	C1, C2, C3, i-C4, n-C4, i-C5, n-C5, C6, Nitrogen, CO2, H2S

2. Create a stream named **Inlet** in the main Simulation Environment and define it as follows:

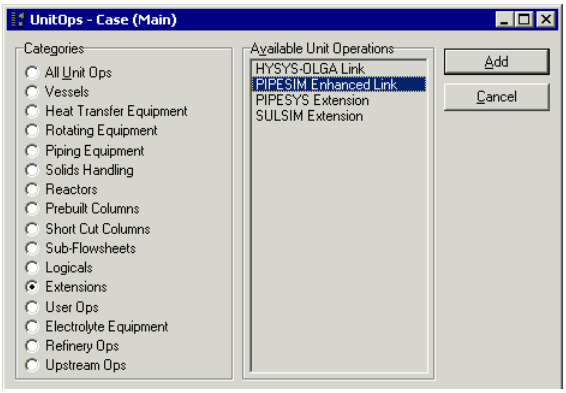
Name	Inlet
Temperature [°C]	150
Pressure [kPa]	16000
Molar Flow [kgmole/h]	360
Comp Mole Frac [methane]	0.7540
Comp Mole Frac [ethane]	0.1696
Comp Mole Frac [propane]	0.0410
Comp Mole Frac [i-Butane]	0.0068
Comp Mole Frac [n-Butane]	0.0100
Comp Mole Frac [i-Pentane]	0.0028
Comp Mole Frac [n-Pentane]	0.0026
Comp Mole Frac [n-Hexane]	0.0060
Comp Mole Frac [Nitrogen]	0.0066
Comp Mole Frac [CO2]	0.0003
Comp Mole Frac [H2S]	0.0003

5.3.2 Adding the PIPESIM Link Extension

Once the case is created, the PIPESIM Link Extension can be added.

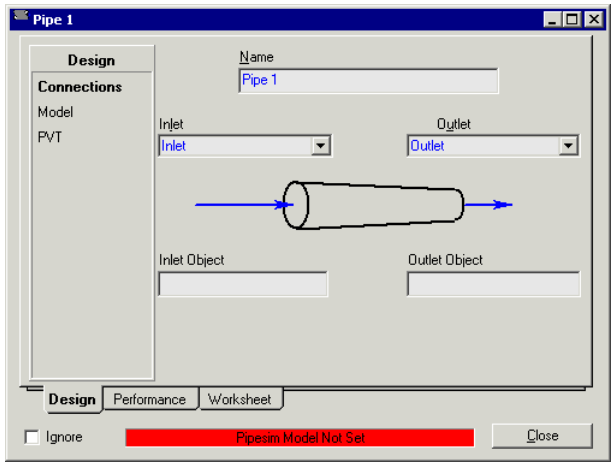
1. From the **Flowsheet** menu, select **Add Operation**. The UnitOps view appears.
2. From the **Categories** group, select the **Extensions** radio button.
3. From the **Available Unit Operations** group, select **PIPESIM Enhanced Link**.

Figure 5.11



4. On the **Connections** page of the **Design** tab complete the page as shown in the figure below.

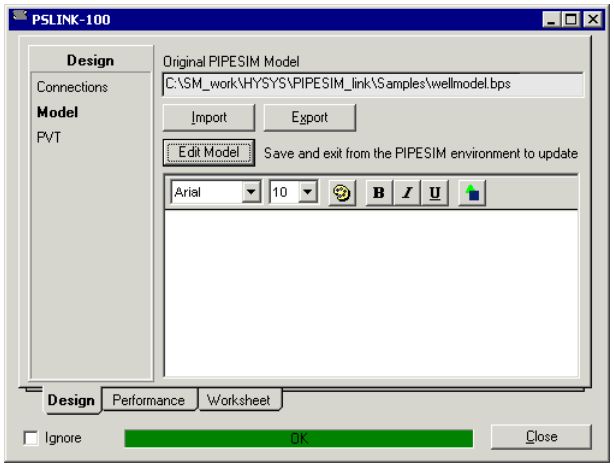
Figure 5.12



5.3.3 Importing the PIPESIM Case

1. Click on the **Model** page on the **Design** tab of the PIPESIM Link property view.
2. Click the **Import** button, and select the location of the PIPESIM model **wellmodel.bps**.
3. To activate and/or to make changes to the PIPESIM model, click the **Edit Model** button.

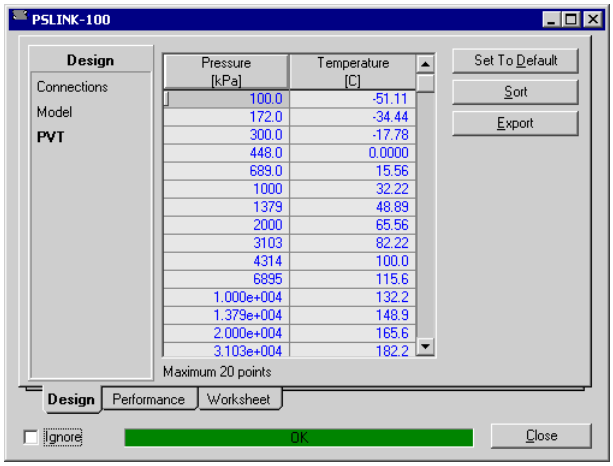
Figure 5.13



For the pipe to solve the PVT physical properties must be specified. A maximum of 20 temperature and 20 pressure points can be specified or defaults pressure and temperature points that correspond to the internal PIPESIM can be selected by clicking the Set to Default button.

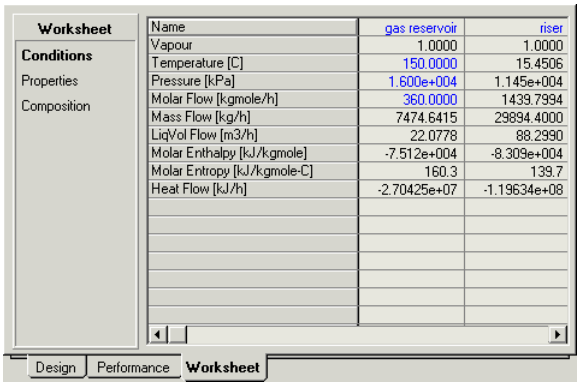
4. Click the **Set To Default** button on the **PVT** page of the **Design** Tab.

Figure 5.14



5. The HYSYS case will run and the stream results will appear as shown in the figure below.

Figure 5.15

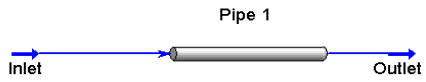


6. Save your completed case as **PIPESIM1.hsc**.

The PFD generated for the completed case, plus a material stream table is shown below:

To add a table to a PFD, right-click on the PFD and select Add Workbook Table command from the Object Inspect menu.

Figure 5.16

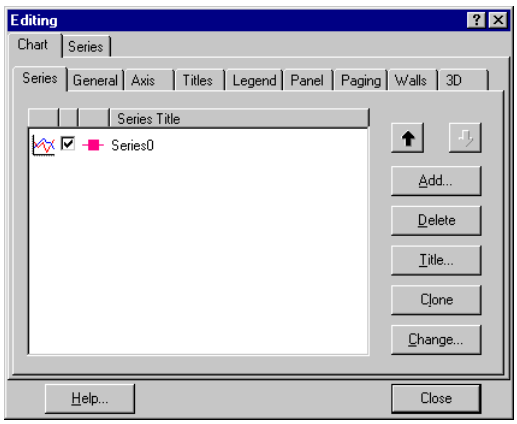


Material Streams			
		Inlet	Outlet
Vapour Fraction		1.0000	1.0000
Temperature	C	150.0	15.45
Pressure	kPa	1.600e+004	1.145e+004
Molar Flow	kgmole/h	360.0	1440
Mass Flow	kg/h	7475	2.989e+004
Liquid Volume Flow	m3/h	22.08	88.30
Heat Flow	kJ/h	-2.704e+007	-1.196e+008

5.3.4 Plotting PIPESIM Data

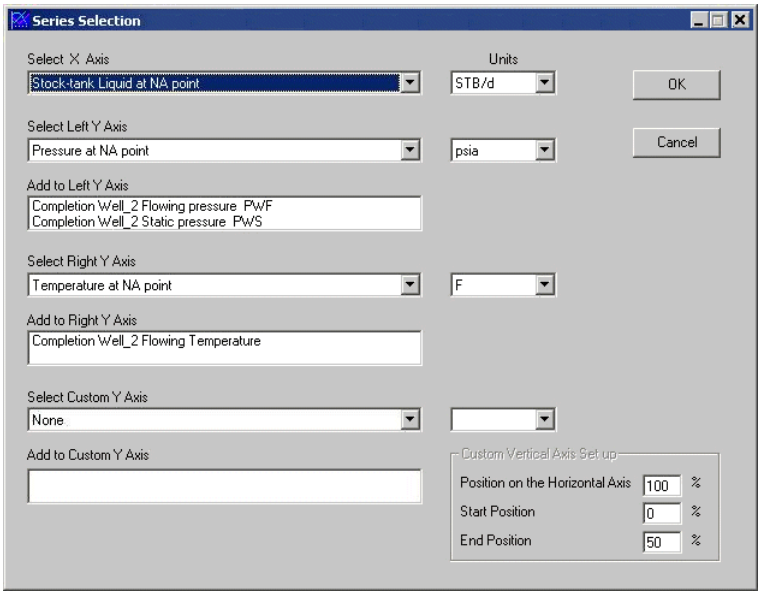
1. On the **Performance** tab, select the **PIPESIM** page.
2. Click the **PSPLOT** button to view a plot of your PIPESIM data. When your plot opens you will see a plot of pressure vs. time.
3. From the **Edit** menu, select **Plot Setup** to add temperature to your plot.

Figure 5.17



4. Click the **Add** button to add a new series to your plot. Select the **Line** plot type and click the **OK** button.
5. Click **Close** to exit the Plot Setup view.
6. Click the **Series** menu to assign data to your new series.

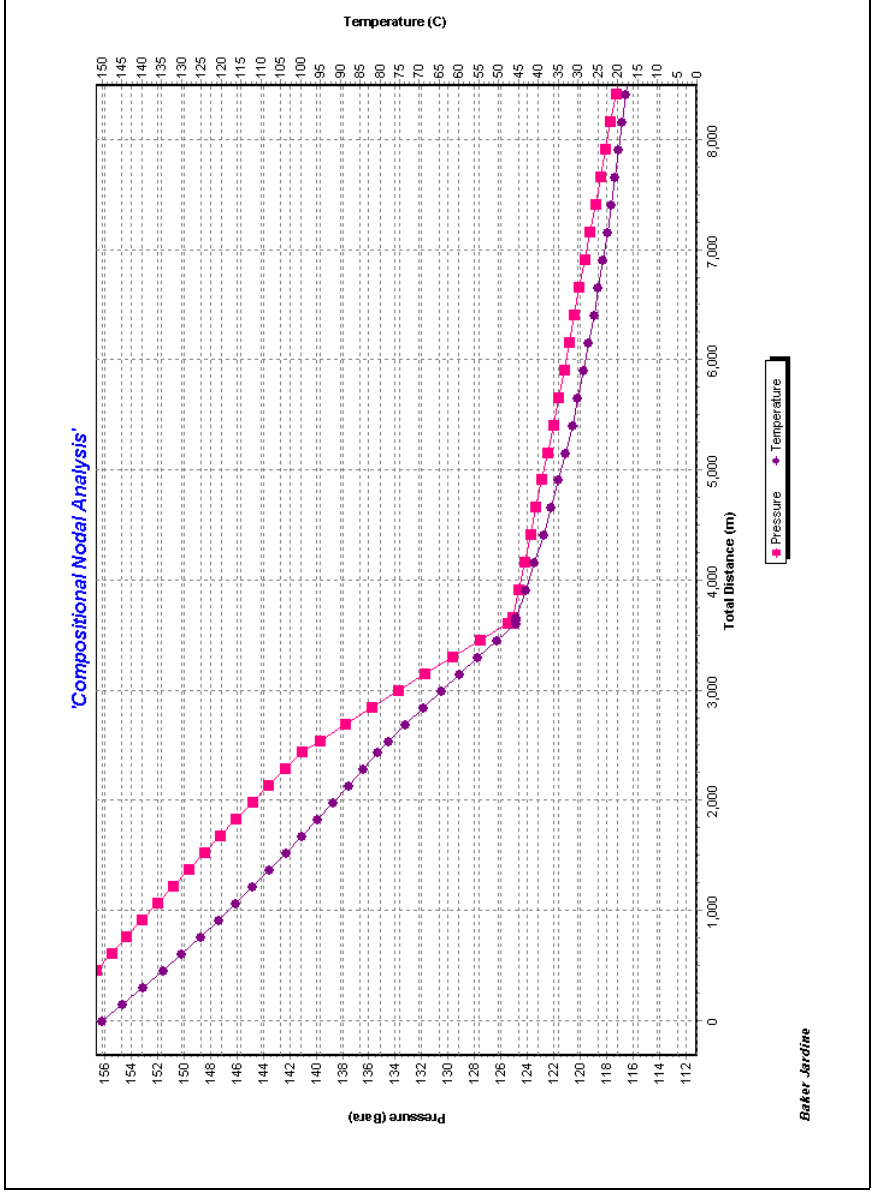
Figure 5.18



7. From the **Select Right Y Axis** drop-down list, select **Temperature (C)** and then click the **OK** button.

When you are finished making these changes your plot will look like the figure below.

Figure 5.19



6 PIPESIM NET

6.1 Introduction

PIPESIM NET is a data model and solver used for the solution of network models. HYSYS links to compositional PIPESIM NET models.

HYSYS also links to the black oil PIPESIM NET models, which allow modeling of Gas Lift systems and faster execution speed for the PIPESIM NET models.

For information on the PIPESIM 2000 Open Link, refer to the Baker Jardine **PIPESIM Reference** manual.

The PIPESIM 2000 GUI is used for editing and calculating PIPESIM NET models.

Open Link is an ActiveX component that allows programmatic access to data within and execution control of the PIPESIM NET models.

6.2 PIPESIM NET

PIPESIM NET is a unit operation for using the PIPESIM software package used to simulate pipeline systems within the HYSYS framework.

HYSYS and PIPESIM 2000 must be installed prior to adding the PIPESIM NET unit operation.

6.2.1 PIPESIM NET Property View

You can also add the PIPESIM NET by clicking the **F12** hot key.

There are two ways that you can add a PIPESIM NET to your simulation:

1. From the **Flowsheet** menu, click **Add Operation**. The UnitOps view appears.
2. Click the **Upstream Ops** radio button.
3. From the list of available unit operations, select **PIPESIM**.
4. Click the **Add** button.

OR

You can also open the Object Palette by clicking the **F4** hot key.

1. From the **Flowsheet** menu, click **Palette**. The Object Palette appears.
2. Click on the **Upstream Ops** icon. The Upstream Palette appears.

Upstream Ops icon

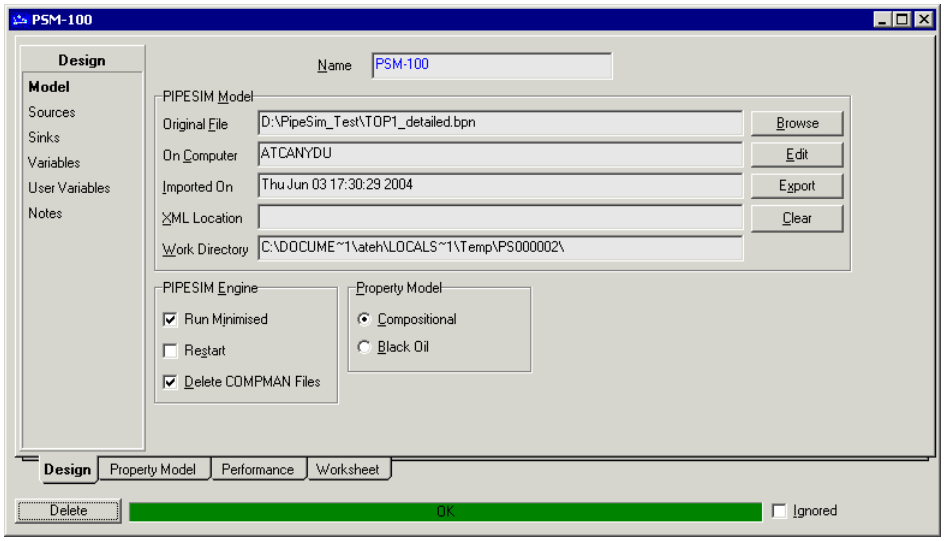
Figure 6.1

3. Double-click the **PIPESIM NET** icon.

PIPESIM NET icon

The PIPESIM property view appears.

Figure 6.2



You can also delete the PIPESIM by clicking on the PIPESIM icon on the PFD and pressing the **DELETE** key.

- To delete the PIPESIM operation, click the **Delete** button. HYSYS will ask you to confirm the deletion.
- To ignore the PIPESIM during calculations, activate the **Ignored** checkbox. HYSYS completely disregards the operation (and cannot calculate the outlet stream) until you restore it to an active state by deactivating the checkbox.

6.2.2 Design Tab

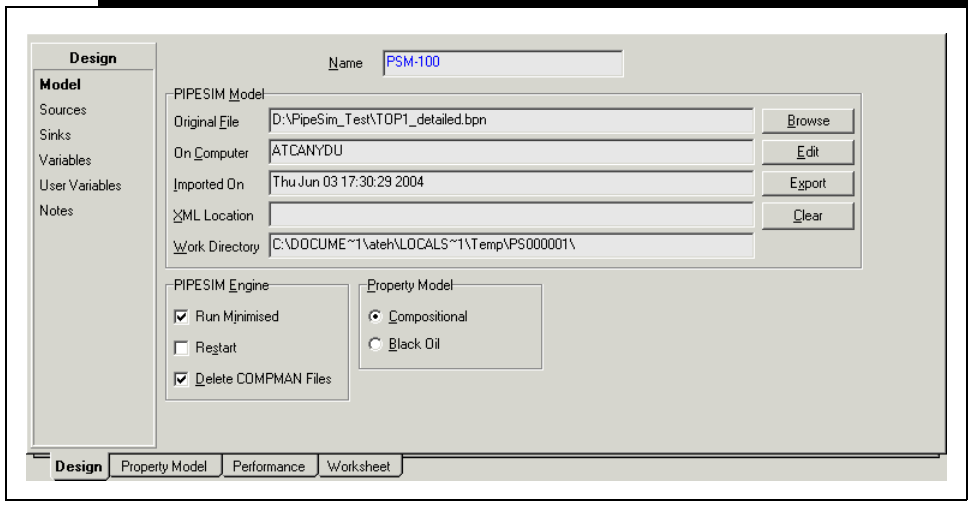
The Design tab consists of the following pages:

- Model
- Sources
- Sinks
- Variables
- User Variables
- Notes

Model Page

The Model page allows you to link to the PIPESIM NET model. The name of the operation can be changed in the **Name** field.

Figure 6.3



The Model page consists of three groups:

- PIPESIM Model
- PIPESIM Engine
- Property Model

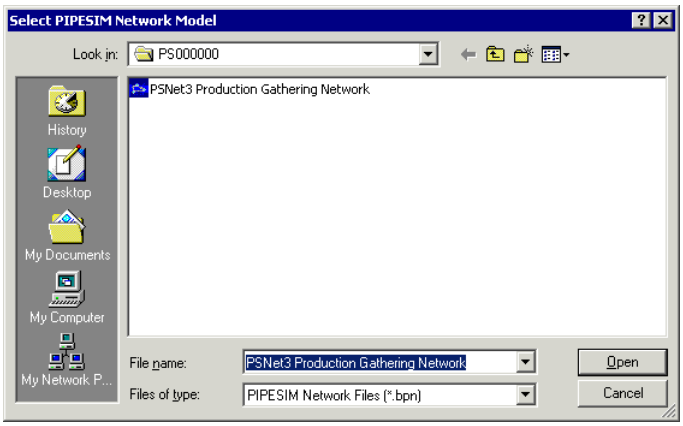
PIPESIM Model Group

The PIPESIM Model group contains options than enables you to configure the location of files used by PIPESIM Net.

- The **Original File**, **On Computer**, and **Imported On** fields are read only fields that display the imported PIPESIM NET model information.
- The **XML Location** field displays the location for the PIPESIM NET model files, if the HYSYS case is managed as an XML file. If the field is empty then the model files are in the same directory as the HYSYS case. The models files are the *.bpn, *.out, *.sum, and *.pns files for the model plus the *.plc and *.plt files for any PIPESIM NET objects that contain profile information.

- The **Work Directory** field is a read only field that is displayed solely to assist in the diagnosis of computer system related problems such as low free disk space.
- The **Browse** button allows you to select an existing PIPESIM NET model (.bpn file).

Figure 6.4

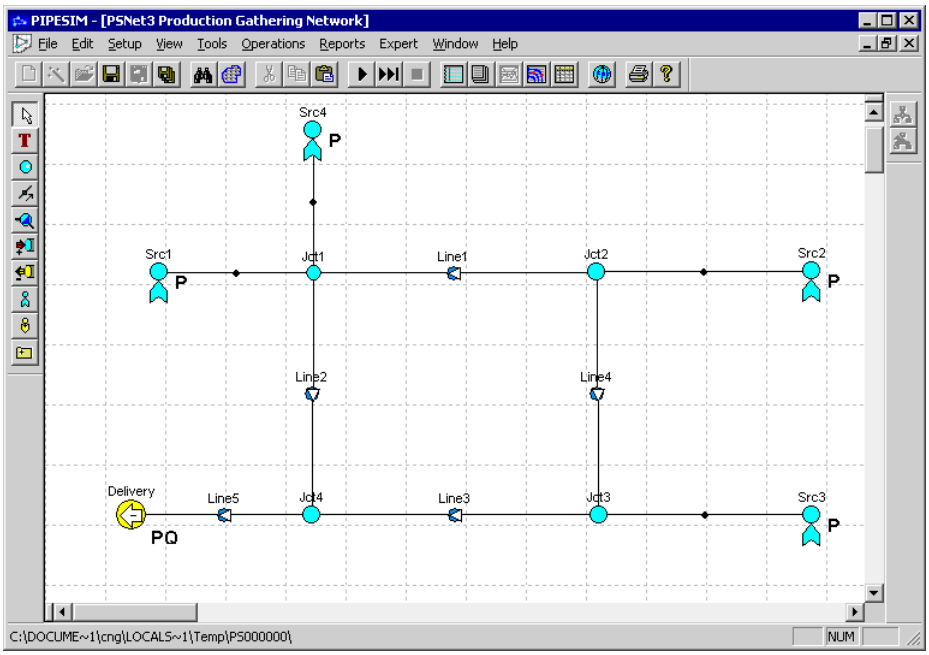


You can select an existing PIPESIM NET model (.bpn file) from the case studies that are available when you have installed PIPESIM. These case studies are located on C:\\Program Files\Schlumberger\PIPESIM\Case Studies.

The selected model is embedded within the HYSYS case, which provides complete portability of the model within a single HYSYS file (.hsc). During a HYSYS session, temporary copies of the files that comprise the model are automatically managed in the Work Directory.

- The **Edit** button allows you to edit the PIPESIM NET model once it has been embedded within the HYSYS case. It modally launches the PIPESIM 2000 GUI and automatically loads the PIPESIM NET model.

Figure 6.5



Once you have finished editing the PIPESIM NET model, you can save it from within the PIPESIM GUI then exit the PIPESIM GUI. The PIPESIM NET model file modification date/time is checked against the pre-edit value to determine if any changes have been made. The model name must not be changed by Save...As when saving the model within PIPESIM 2000.

- The **Export** button allows you to export the embedded PIPESIM NET model file.
- The **Clear** button clears the contents of the embedded model. This works by embedding the empty template model file newmodel.bpn located in the HYSYS \support subdirectory. This is the state of the link when the instance is first created.

PIPESIM Engine Group

In the PIPESIM Engine group there are three checkboxes:

The PIPESIM Engine is controlled by the Run Minimised and Restart checkboxes.

Checkbox	Description
Run Minimised	Run Minimised causes the PIPESIM NET engine process to open minimised on the task bar rather than full screen.
Restart	Restart causes the PIPESIM NET engine to initialise the new solution from the last solution, useful for case studies, optimisation and recycle calculations.
Delete COMPMAN Files	Deletes the compositional work files created by PIPESIM NET in the Windows Temporary directory, each time the link is executed.

Property Model Group

From the Property Model group, contains two radio buttons that enables you to toggle between the following property model:

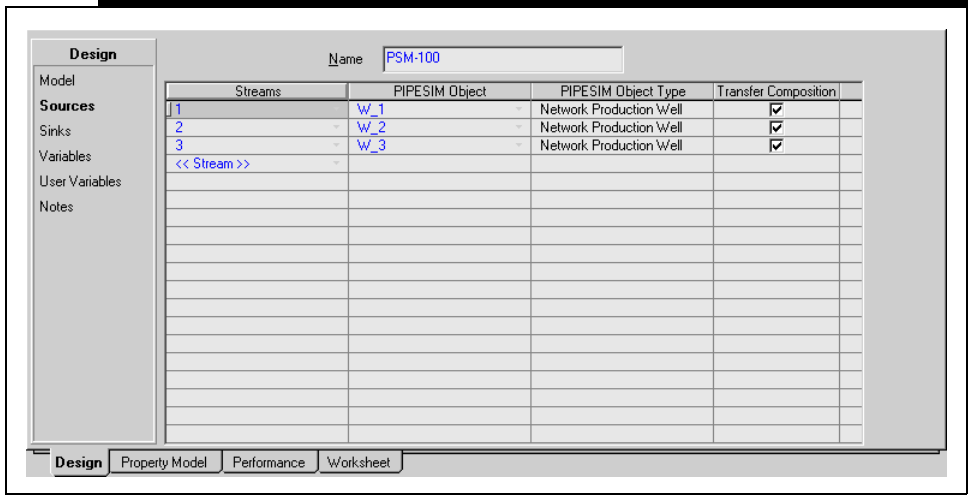
- Compositional
- Black Oil

When creating a new/empty *.bpn model, the default property model is Compositional.

Sources Page

The Sources page allows you to specify the feed stream and PIPESIM object.

Figure 6.6



When you have selected a feed stream from the **Streams** column drop-down list, you can also select the PIPESIM object from the **PIPESIM Object** column drop-down list. The **PIPESIM Object Type** column shows the type of object based on the PIPESIM object you selected in the PIPESIM Object column.

If the **Transfer Composition** checkbox is selected as shown in the figure above, then the composition of the feed (compositional or black oil model) is transferred between HYSYS and PIPESIM each time the PIPESIM engine is executed.

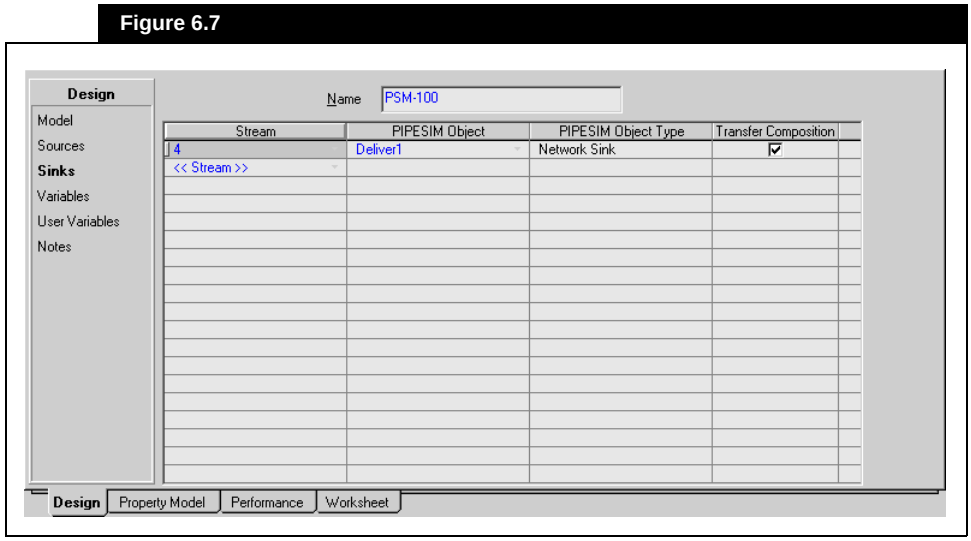
The transfer is forward only (HYSYS to PIPESIM), and the transfer is in addition to any specifications on pressure, temperature, and mass flow. Negative flow rates are not supported.

The PIPESIM model must contain at least one sink pressure specification made either within the PIPESIM model or from HYSYS.

Sinks Page

The Sinks page allows you to specify the product stream, and PIPESIM object.

Figure 6.7



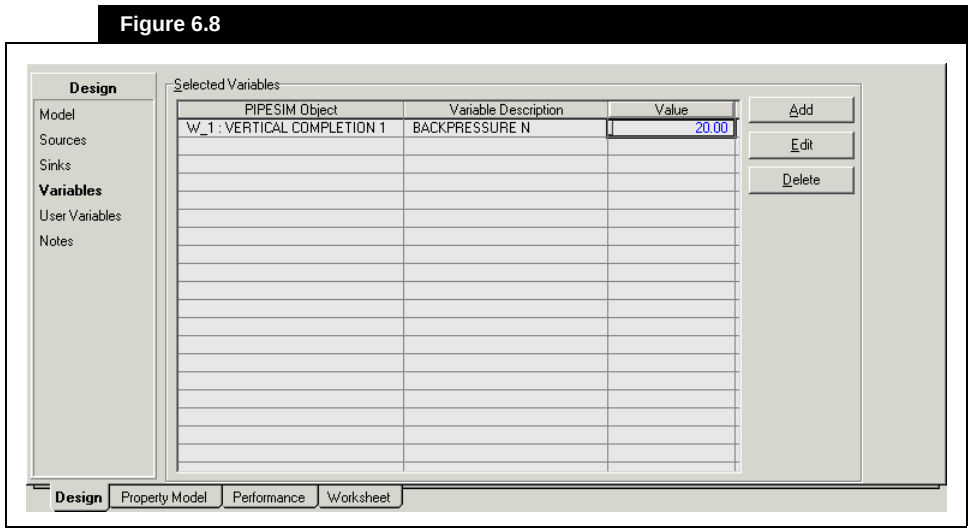
When you have selected a product stream from the **Streams** column drop-down list, you can also select the PIPESIM object from the **PIPESIM Object** column drop-down list. The **PIPESIM Object Type** column shows the type of object based on the PIPESIM object you selected in the PIPESIM Object column.

If the **Transfer Composition** checkbox is selected, then the composition of the product (compositional or black oil model) is transferred between HYSYS and PIPESIM each time the PIPESIM engine is executed.

Variables Page

The Variables page allows you to interact with any PIPESIM NET data variable that is available via the Open Link. The variables have either read or write access with the exception of the depth of sub-equipment within the PIPESIM NET tubing model.

Figure 6.8

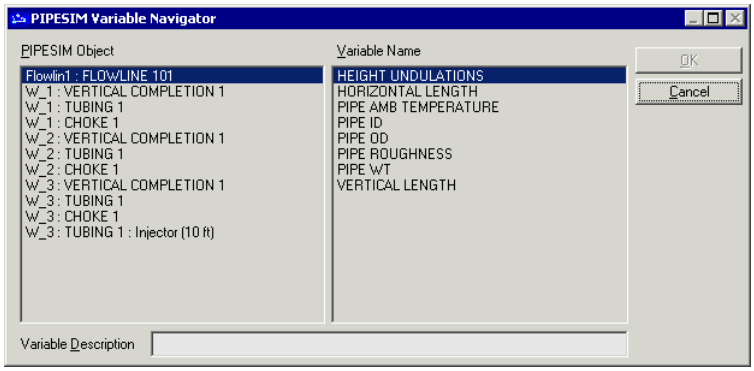


Open Link is an ActiveX component that allows programmatic access to data within and execution control of the PIPESIM NET models.

PIPESIM Variable Navigator View

When you click the **Add** or **Edit** button in the **Variables** page of the PIPESIM Net property view, the PIPESIM Variable Navigator appears.

Figure 6.9



This view contains a list of available PIPESIM objects and the associated variable names. The **Variable Name** list contains a list of all the supported properties for the selected **PIPESIM Object**.

The PIPESIM objects colon ":" indicates a new level within the PIPESIM model. W_1: VERTICAL COMPLETION 1 indicates a single branch unit operation "VERTICAL COMPLETION 1" in network unit operation "W_1".

An additional level can apply for sub-equipment within a tubing unit operation.

In the **Variable Description** field, you can enter the name for each variable to be added to the list of Selected Variables on the **Variables** page of the PIPESIM Net property view.

User Variables Page

For more information on implementing the User Variables, refer to **Chapter 5 - User Variables** in the **HYSYS Customization Guide**.

The User Variables page allows you to create and implement variables in the HYSYS simulation case.

Notes Page

For more information, refer to **Section 7.20 - Notes Manager** in the **HYSYS User Guide**.

The Notes page provides a text editor that allows you to record any comments or information regarding the specific unit operation, or the simulation case in general.

6.2.3 Property Model Tab

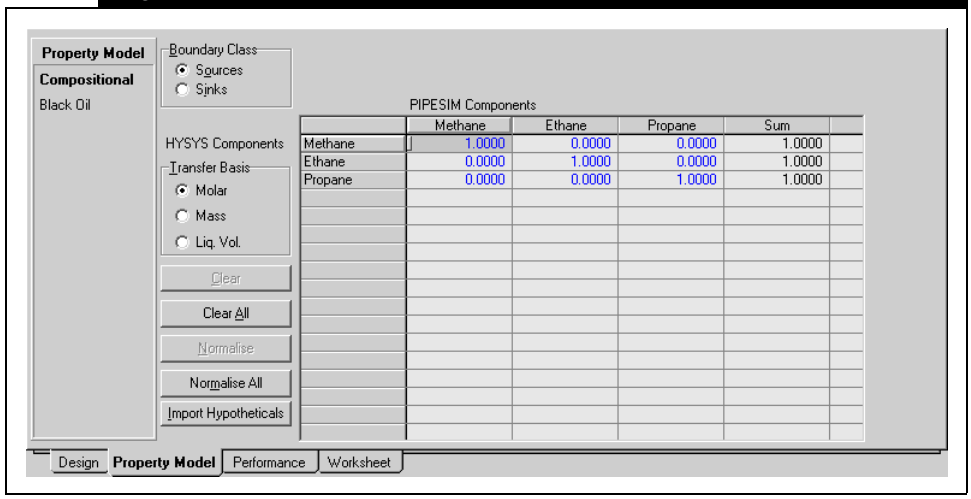
The Property Model tab consists of the following pages:

- Compositional
- Black Oil

Compositional Page

You can map boundary compositions on the Compositional page.

Figure 6.10

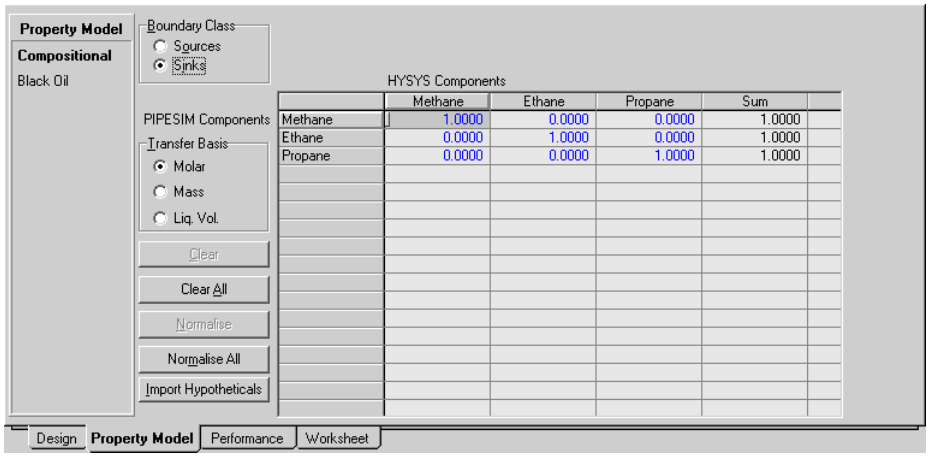


In the Boundary Class group, you can click on the Sources or Sinks radio button to define the component map. Component maps are defined for transfer of the composition between HYSYS and PIPESIM NET which defines the Sources component map, and between PIPESIM NET and HYSYS which defines the Sinks component map.

For the Sources boundary class, the first column of the component mapping table (in the figure above) shows a list of components in the HYSYS flowsheet that contains the PIPESIM NET link unit operation.

For the Sinks boundary class, the first column of the component mapping table shows the list of components in the PIPESIM NET model.

Figure 6.11



The transfer of the composition is based on the selected basis type in the Transfer Basis group. There are three types of basis available: Molar, Mass, or Liq. Vol., each basis is associated to a radio button.

The Compositional page has the following buttons:

Button	Description
Clear	Resets all mapping factors to zero for the selected row.
Clear All	Resets all mapping factors to zero for all rows.
Normalise	Normalises the mapping factors to 1 for the selected row.
Normalise All	Normalises the mapping factors to 1 for all rows.
Import Hypotheticals	Imports the definitions of any hypothetical (petroleum fraction) components into the list of HYSYS components.

When a PIPESIM NET model is first imported into the link, the component maps are automatically initialized where possible. The file pscomps.sdb in the HYSYS\Support subdirectory contains the mapping between HYSYS and PIPESIM 2000 component names.

Black Oil Page

The options in this page is not yet available for the current unit operation.

6.2.4 Performance Tab

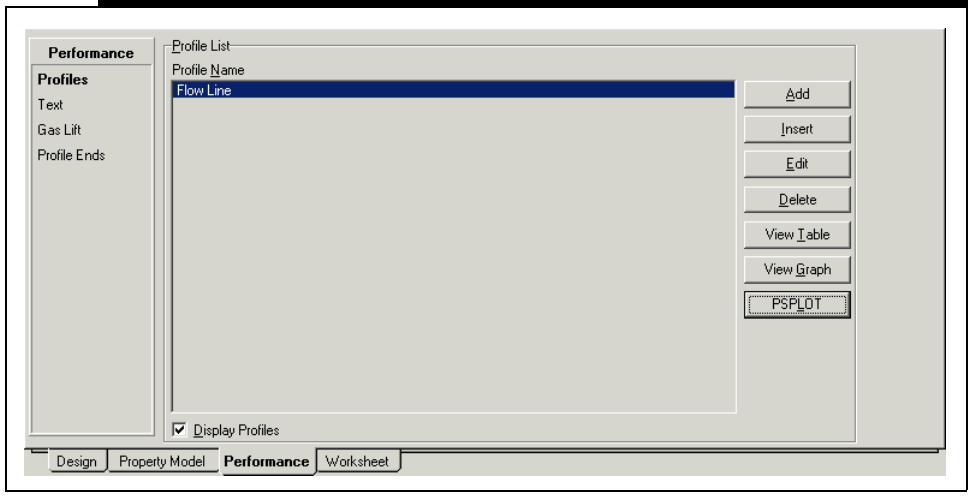
The Performance tab consists of the following pages:

- Profiles
- Text

Profiles page

You can view the PIPESIM NET results on the profiles page.

Figure 6.12



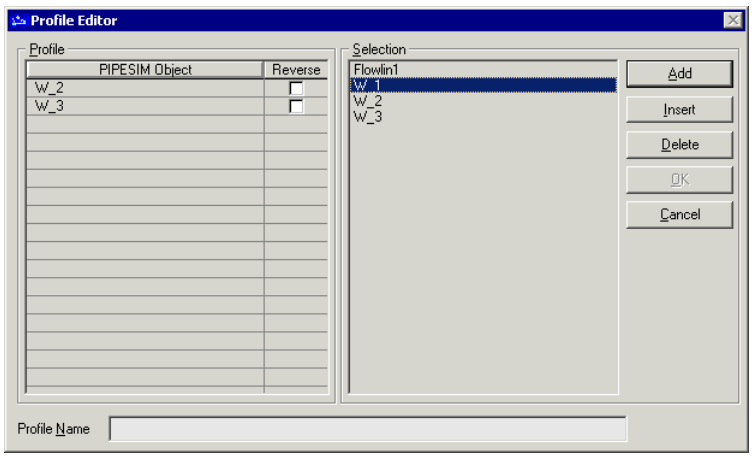
If you want to display profile data, select the Display Profiles checkbox. The extraction of profile data from the PIPESIM NET output files is a slow operation. Disabling the display of profile data during recycle, optimization or case study calculations can significantly increase the speed of execution. The profiles can be displayed after the calculations are complete.

Adding and Editing Profiles

1. In the **Profiles** page, click the **Add** button to add a profile or click the **Edit** button to edit a profile.

   The Profile Editor appears.

Figure 6.13



   The **Add** button adds to the Profile List after the currently selected profile, whereas the **Insert** button adds to the Profile List before the currently selected profile.

2. From the Selection group, select the PIPESIM NET unit operation you want to add to the profile.
3. Click the **Add** or **Insert** button.

   The **Add** button adds to the PIPESIM Object list after the currently selected PIPESIM Object, whereas the **Insert** button adds to the PIPESIM Object list before the currently selected PIPESIM Object.

4. The selected PIPESIM NET unit operation appears in the PIPESIM Object list of the Profile group.

   If you want to remove the PIPESIM Object from the profile, select the PIPESIM Object you want to remove and click the **Delete** button.

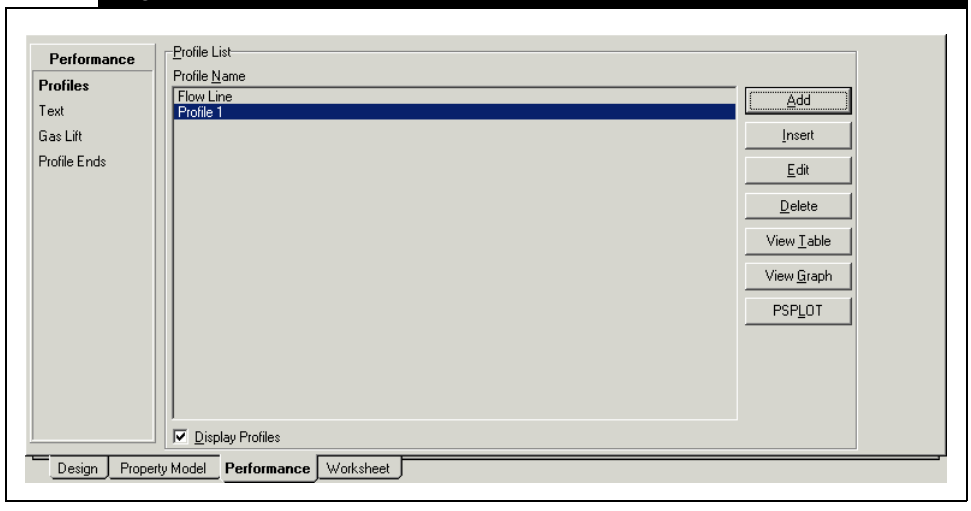
   You can select the **Reverse** checkbox if you want to reverse the profile information for the PIPESIM Object. This is required to correctly display the profile when you have reverse flow through the PIPESIM Object.

5. In the **Profile Name** field, type the name for the profile and press **ENTER**.

   The profile name appears in the Profile List of the **Profiles** page.

6. Click the **OK** button to return to the **Profiles** page.

Figure 6.14



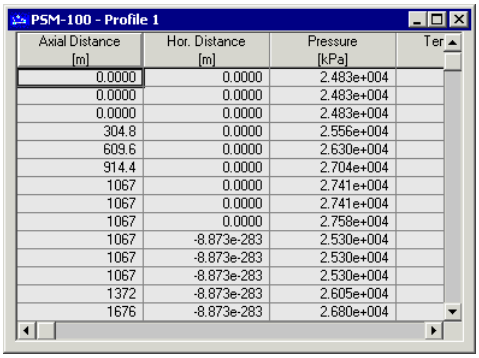
If you want to remove the profile name from the Profile List, select the profile name and click the **Delete** button.

Viewing Profiles

You can define composite profiles that contain multiple network unit operations, which allows you to view the profile between the source and the sink.

When you click the **View Table** button of the Profiles page, a tabular view of the calculated data for the selected profile appears. The figure below shows an example of a tabular view.

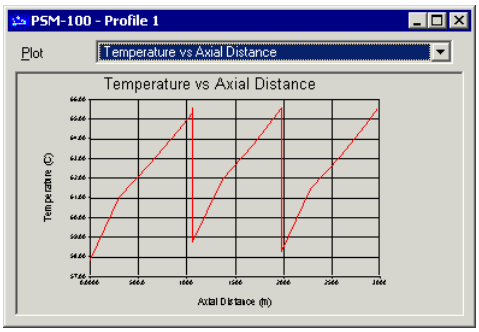
Figure 6.15



Axial Distance [m]	Hor. Distance [m]	Pressure [kPa]	Ter
0.0000	0.0000	2.483e+004	
0.0000	0.0000	2.483e+004	
0.0000	0.0000	2.483e+004	
304.8	0.0000	2.556e+004	
609.6	0.0000	2.630e+004	
914.4	0.0000	2.704e+004	
1067	0.0000	2.741e+004	
1067	0.0000	2.741e+004	
1067	0.0000	2.758e+004	
1067	-8.873e-283	2.530e+004	
1067	-8.873e-283	2.530e+004	
1067	-8.873e-283	2.530e+004	
1372	-8.873e-283	2.605e+004	
1676	-8.873e-283	2.680e+004	

When you click the **View Graph** button of the Profiles page, a graphical view of the calculated data for a specific variable for the selected profile appears. The figure below shows an example of a graphical view.

Figure 6.16

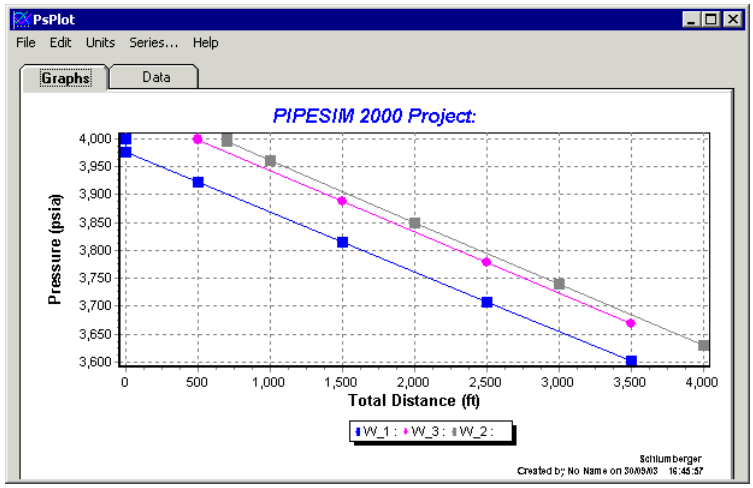


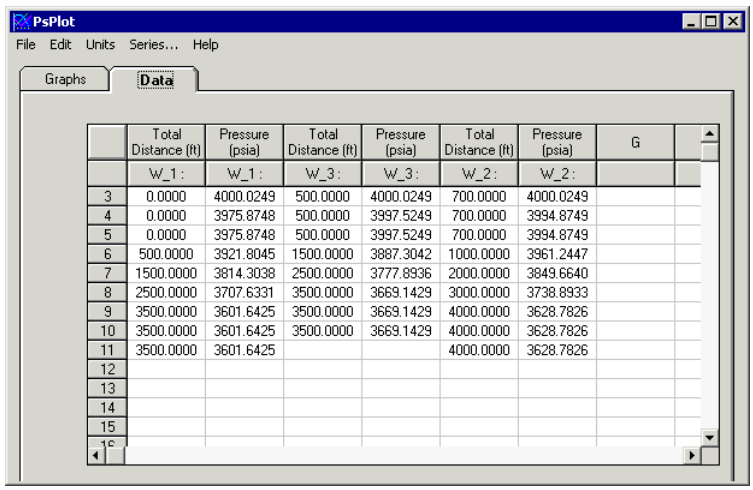
Launching PSPLOT

PSPLOT allows you to view the calculated profiles for the unit operation within the PIPESIM NET models. To launch PSPLOT, click the **PSPLOT** button of the Profiles page.

You can view the profiles as a graph or table by clicking on the Graph or Data tab of the PSPLOT program.

Figure 6.17



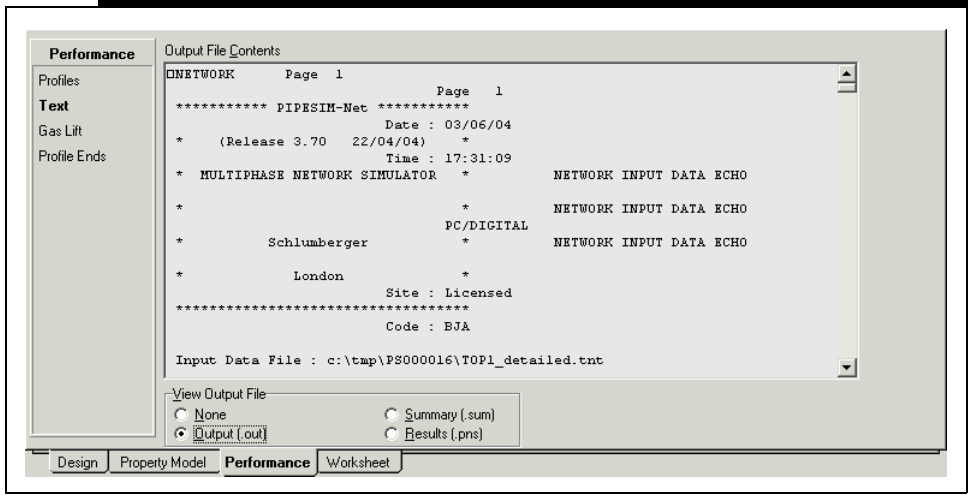


	Total Distance (ft)	Pressure (psia)	Total Distance (ft)	Pressure (psia)	Total Distance (ft)	Pressure (psia)	G
	W_1 :	W_1 :	W_3 :	W_3 :	W_2 :	W_2 :	
3	0.0000	4000.0249	500.0000	4000.0249	700.0000	4000.0249	
4	0.0000	3975.8748	500.0000	3997.5249	700.0000	3994.8749	
5	0.0000	3975.8748	500.0000	3997.5249	700.0000	3994.8749	
6	500.0000	3921.8045	1500.0000	3887.3042	1000.0000	3961.2447	
7	1500.0000	3814.3038	2500.0000	3777.8936	2000.0000	3849.6640	
8	2500.0000	3707.6331	3500.0000	3669.1429	3000.0000	3738.8933	
9	3500.0000	3601.6425	3500.0000	3669.1429	4000.0000	3628.7826	
10	3500.0000	3601.6425	3500.0000	3669.1429	4000.0000	3628.7826	
11	3500.0000	3601.6425			4000.0000	3628.7826	
12							
13							
14							
15							

Text Page

On the Text page you can view the PIPESIM NET output files content.

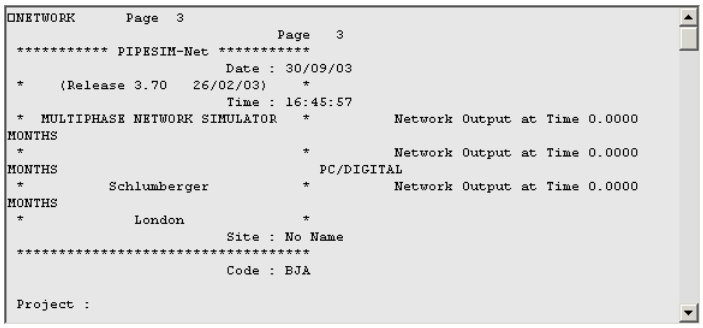
Figure 6.18



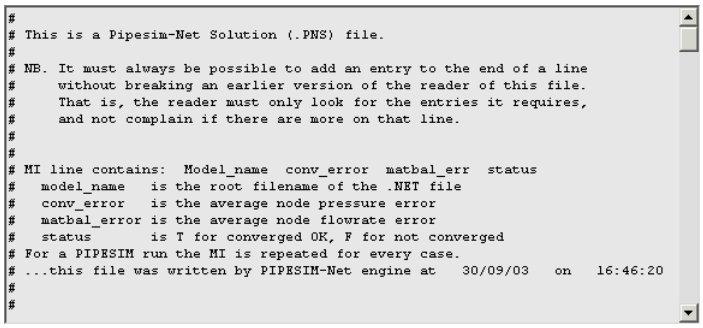
In the View Output File group, the radio buttons allow you to display the contents of the results files. The are four radio buttons: None, Output (.out), Summary (.sum) or Results(.pns). The figure above shows the output file contents when the Output radio button is selected.

The figure below shows the Summary and Results output file contents.

Figure 6.19



Summary Output File Contents



Results Output File Contents

If processor/calculation speed is of importance, you can select the **None** radio button because the Output File contents are updated after every calculation.

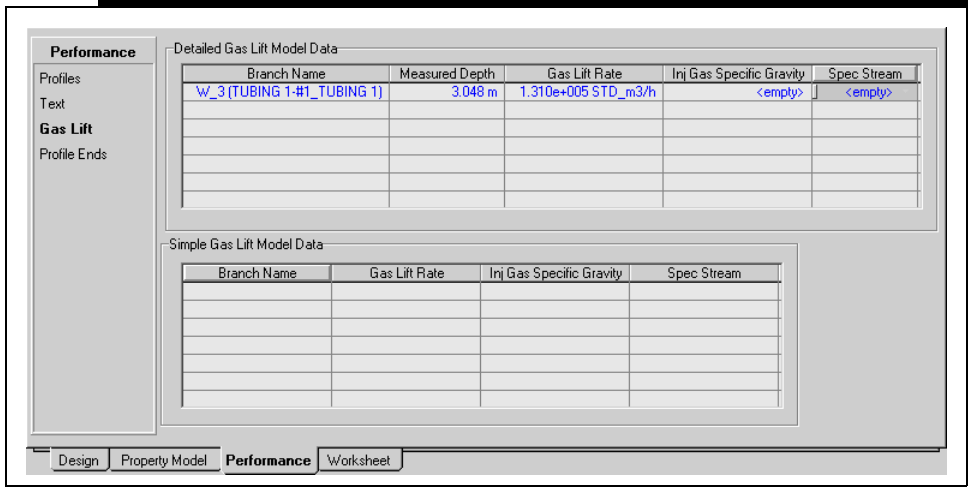
You can print the contents of the displayed output file by right-clicking on the output file contents. The object inspect menu appears.

Figure 6.20

Gas Lift Page

The Gas Lift page enables you to configure the Detailed or Simple Gas Lift model.

Figure 6.21

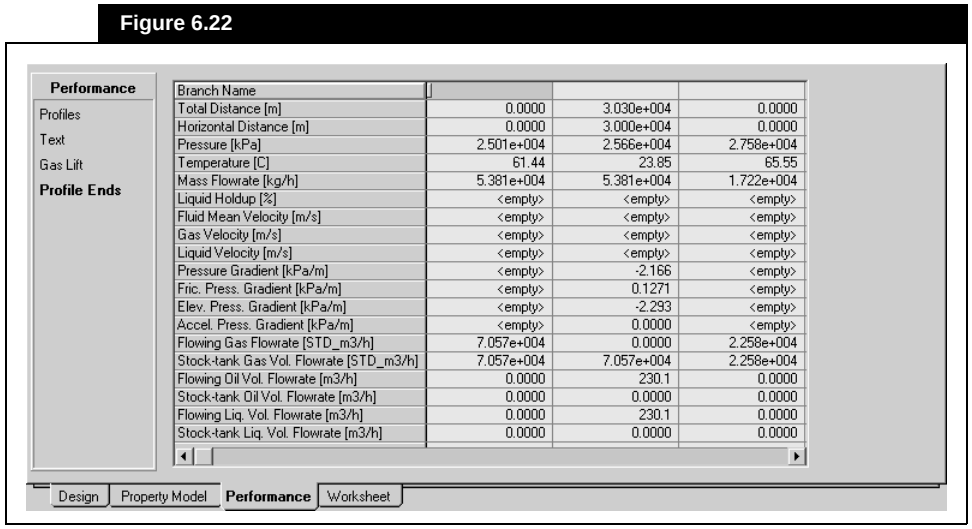


For more information on the variable values displayed in the Gas Lift page, refer to the Baker Jardine **PIPESIM Reference** manual.

Profile Ends Page

The Profile Ends page displays the profile end properties of the PIPESIM Net operation.

Figure 6.22



Branch Name			
Total Distance [m]	0.0000	3.030e+004	0.0000
Horizontal Distance [m]	0.0000	3.000e+004	0.0000
Pressure [kPa]	2.501e+004	2.566e+004	2.758e+004
Temperature [C]	61.44	23.85	65.55
Mass Flowrate [kg/h]	5.381e+004	5.381e+004	1.722e+004
Liquid Holdup [%]	<empty>	<empty>	<empty>
Fluid Mean Velocity [m/s]	<empty>	<empty>	<empty>
Gas Velocity [m/s]	<empty>	<empty>	<empty>
Liquid Velocity [m/s]	<empty>	<empty>	<empty>
Pressure Gradient [kPa/m]	<empty>	-2.166	<empty>
Fric. Press. Gradient [kPa/m]	<empty>	0.1271	<empty>
Elev. Press. Gradient [kPa/m]	<empty>	-2.293	<empty>
Accel. Press. Gradient [kPa/m]	<empty>	0.0000	<empty>
Flowing Gas Flowrate [STD_m3/h]	7.057e+004	0.0000	2.258e+004
Stock-tank Gas Vol. Flowrate [STD_m3/h]	7.057e+004	7.057e+004	2.258e+004
Flowing Oil Vol. Flowrate [m3/h]	0.0000	230.1	0.0000
Stock-tank Oil Vol. Flowrate [m3/h]	0.0000	0.0000	0.0000
Flowing Liq. Vol. Flowrate [m3/h]	0.0000	230.1	0.0000
Stock-tank Liq. Vol. Flowrate [m3/h]	0.0000	0.0000	0.0000

6.2.5 Worksheet Tab

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the unit operation. The Conditions and Composition pages contain selected information from the corresponding pages of the Worksheet tab for the stream property view.

The Properties page displays the property correlations of the inlet and outlet streams of the unit operations. The following is a list of the property correlations:

- Vapour / Phase Fraction
- Temperature
- Pressure
- Actual Vol. Flow
- Mass Enthalpy
- Mass Entropy
- Molecular Weight
- Vap. Frac. (molar basis)
- Vap. Frac. (mass basis)
- Vap. Frac. (volume basis)
- Molar Volume
- Act. Gas Flow
- Act. Liq. Flow
- Std. Liq. Flow

- Molar Density
- Mass Density
- Std. Ideal Liquid Mass Density
- Liquid Mass Density
- Molar Heat Capacity
- Mass Heat Capacity
- Thermal Conductivity
- Viscosity
- Surface Tension
- Specific Heat
- Z Factor
- Std. Gas Flow
- Watson K
- Kinematic Viscosity
- Cp/Cv
- Lower Heating Value
- Mass Lower Heating Value
- Liquid Fraction
- Partial Pressure of CO2
- Avg. Liq. Density
- Heat of Vap.
- Mass Heat of Vap.

The Heat of Vapourisation for a stream in HYSYS is defined as the heat required to go from saturated liquid to saturated vapour.

7 GAP

7.1 Introduction

The GAP unit operation provides a link between HYSYS and Petroleum Experts.

With this operation, you can import a GAP simulation model, export streams from a HYSYS simulation case into the GAP model, perform calculations and use the capabilities of the GAP model to generate product streams, and import the product streams back into the HYSYS simulation case for further analysis or processing.

In essence, the GAP unit operation acts as a black box using streams and fluids to characterize the input and output boundaries of the GAP model from Petroleum Experts software.

The GAP unit operation is for advance users of Petroleum Experts software. Refer to the Reference manuals provided by the Petroleum Experts for detailed information on using GAP models.

7.2 GAP Property View

There are two methods to add a Petroleum Experts GAP to your simulation:

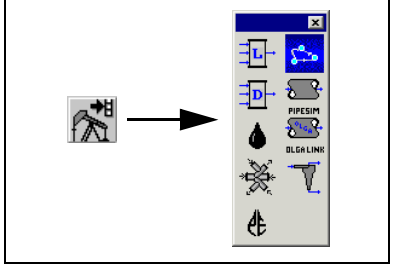
You can also access the UnitOps view by pressing **F12**.

1. From the **Flowsheet** menu, click **Add Operation**. The UnitOps view appears.
2. Click the **Upstream Ops** radio button.
3. From the list of available unit operations, select **Petroleum Experts GAP**.
4. Click the **Add** button.

OR

1. From the **Flowsheet** menu, select **Palette** (or press **F4**). The Object Palette appears.
2. In the Object Palette, click the **Upstream Ops** icon to open the Upstream Object Palette.

Figure 7.1

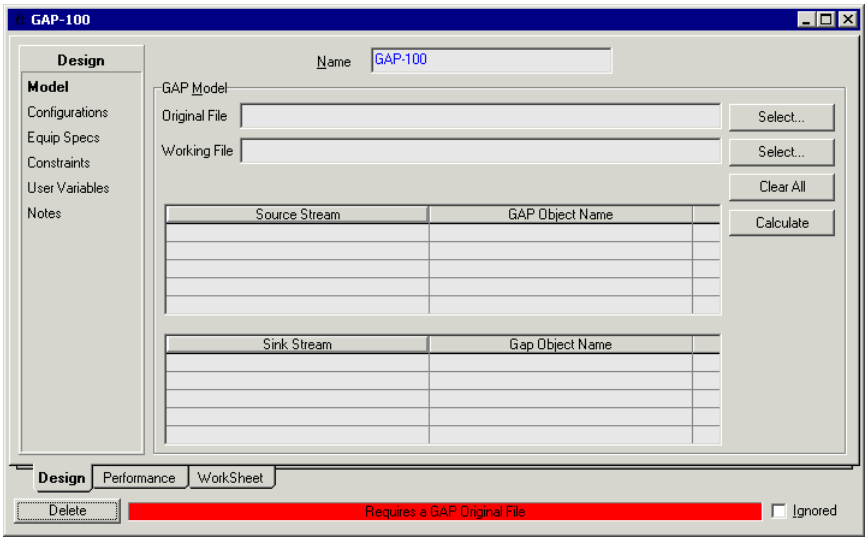


Petroleum Experts GAP icon

3. In the Upstream Object Palette, double-click the **Petroleum Experts GAP** icon.

The GAP property view appears.

Figure 7.2



You can also delete a GAP operation by clicking on the GAP icon on the PFD and pressing **DELETE**.

To delete the GAP operation, click the **Delete** button. HYSYS will ask you to confirm the deletion.

To ignore the GAP during calculations, select the **Ignored** checkbox. HYSYS completely disregards the operation (and cannot calculate the outlet stream) until you restore it to an active state by clearing the checkbox.

7.2.1 Design Tab

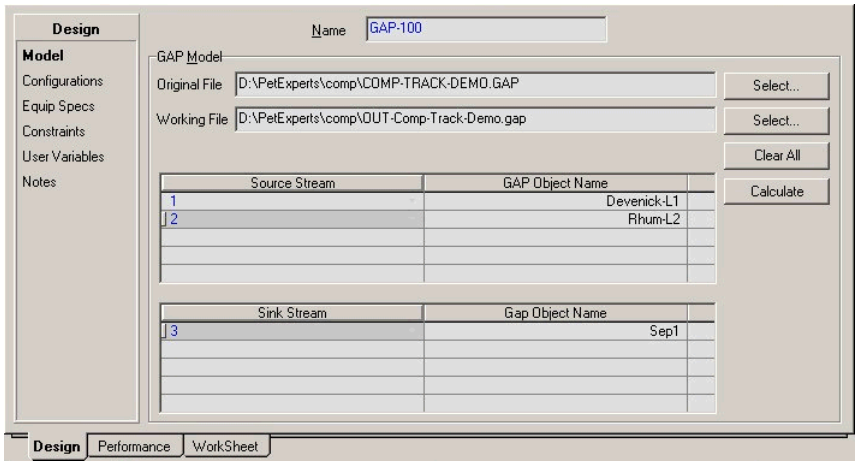
The Design tab consists of the following pages:

- Model
- Configurations
- Equip Specs
- User Variables
- Notes

Model Page

The Model page is used to select the GAP model and define all of the stream connections to the objects in the selected GAP model.

Figure 7.3



The following table lists and describes each option in the Model page:

Object	Description
Name field	Enables you to modify the name of the GAP operation.
Original File field	Displays the location and name of the original GAP model attached to the operation. The imported GAP file cannot be modified.
Select button	Enables you to find and select the original GAP model file.
Working File field	Displays the location and name of the working GAP model attached to the operation. When you make modifications to the GAP model attached to HYSYS, a working GAP model (containing the changes) is created. The purpose of this feature is to enable you to make modifications to the copy of the imported GAP file.
Select button	Enables you to save the working GAP model file.
Clear All button	Enables you to clear all data model and stream attachments to the GAP operation.
Calculate button	Enables you to propagate the calculated results from the GAP model back into the HYSYS simulation case.
Source Stream column	Enables you to select and connect source/inlet streams from HYSYS simulation case to objects in the selected GAP model.
GAP Object Name column	Displays the available inlet streams from the objects in the selected GAP model.
Sink Stream column	Enables you to select and connect sink/outlet streams from HYSYS simulation case to objects in the selected GAP model.
GAP Object Name column	Displays the available outlet streams from the objects in the selected GAP model

Configurations Page

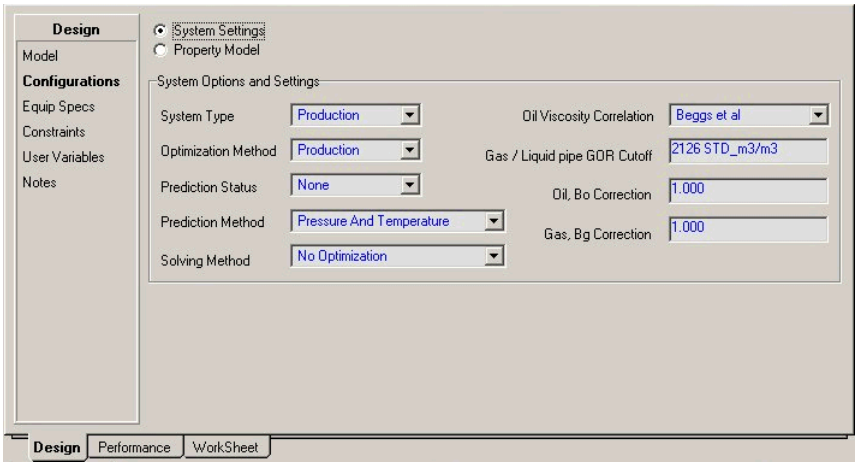
The Configurations page allows you to configure the Petroleum Experts GAP operation. There are two configuration options for the GAP operation: System Settings or Property Model.

The Property Model configuration is only applicable to a GAP model that is Compositional (in other words, not BlackOils).

Depending on which configuration option you selected, the following variables are available for you to configure the GAP operation:

- If you select **System Settings** radio button.

Figure 7.4



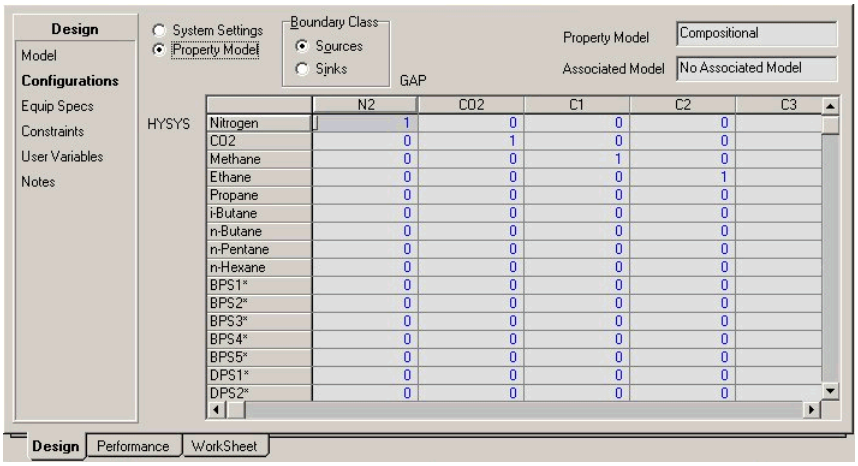
The following variables are available:

Object	Description
System Type drop-down list	Enables you to select the type of system for the GAP operation. You have the following selection: • Production • Water Injection • Gas Injection
Optimization Method drop-down list	Enables you to select the optimization method for the GAP operation. You have the following selection: • Production • Revenue • Oil Rate Only • Gas Rate Only • Water Rate Only
Prediction Status drop-down list	Enables you to toggle between activating (select **On**) or ignoring (select **None**) the selected prediction method.
Prediction Method drop-down list	Enables you to select the prediction method for the GAP operation. You have the following selection: • Pressure Only • Pressure And Temperature
Solving Method drop-down list	Enables you to select the solving method for the GAP operation. You have the following selection: • No Optimization • Optimization With Constraints • Optimization WithOut Constraints
Oil Viscosity Correlation drop-down list	Enables you to select the correlation for the oil viscosity in the GAP operation. You have the following selection: • Beal et al • Beggs et al • Petrosky et al
Gas / Liquid pipe GOR Cutoff field	Enables you to specify the gas liquid ratio value of the pipe gas oil ratio cutoff.
Oil, Bo Correction field	Enables you to specify the Bo correction value for the oil calculation.
Gas, Bg Correction field	Enables you to specify the Bg correction value for the gas calculation.

The selected optimization method is not applicable if the selection for the **Solving Method** drop-down list is **No Optimization**.

- If you select **Property Model** radio button.

Figure 7.5



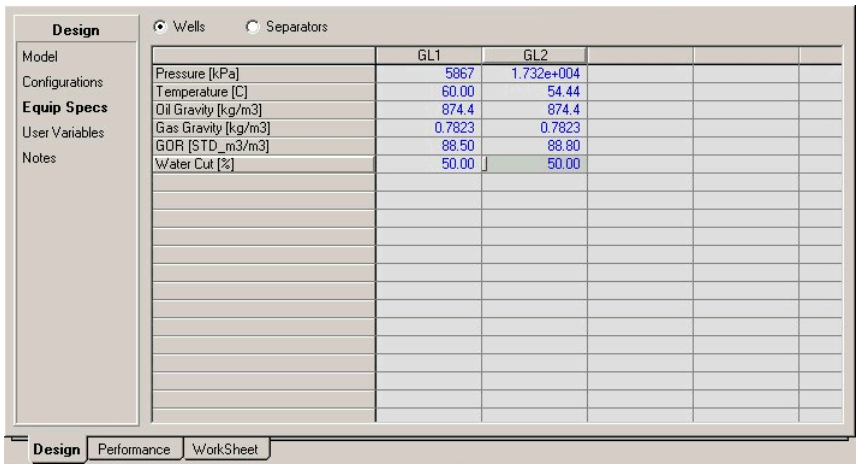
The following variables are available:

Object	Description
Property Model field	Displays the property package of the selected GAP model.
Associated Model field	Displays the name of any models associated to the selected GAP model.
Sources radio button	Enables you to access the HYSYS vs. GAP component table for source/inlet streams.
Sinks radio button	Enables you to access the HYSYS vs. GAP component table for sink/outlet streams.
HYSYS vs. GAP table	Enables you to specify the ratio value of components flowing between the HYSYS and GAP model.

Equip Specs Page

The Equip Specs page allows you to specify GAP object variable values. Depending on the selected GAP model, not all GAP objects will be available for configuration.

Figure 7.6



The radio buttons at the top of the Equip Specs page enables you to select which possible group of GAP objects you can see and modify.

The GAP objects are split into the following groups:

- Wells
- Separators

User Variables Page

The User Variables page allows you to create and implement variables in the HYSYS simulation case. For more information on implementing the User Variables, refer to **Chapter 5 - User Variables** in the **HYSYS Customization Guide**.

Notes Page

For more information, refer to **Section 7.20 - Notes Manager** in the **HYSYS User Guide**.

The Notes page provides a text editor that allows you to record any comments or information regarding the specific unit operation, or the simulation case in general.

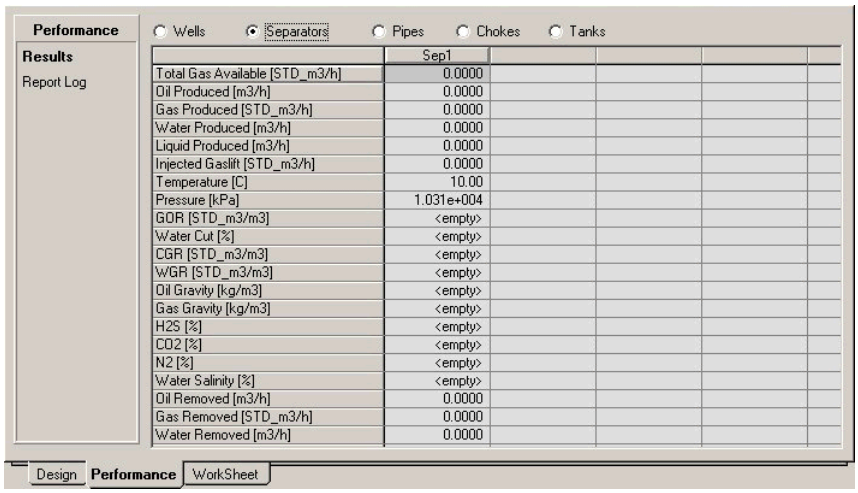
7.2.2 Performance Tab

The Performance tab displays the calculated performance results of the Petroleum Experts GAP operation.

Results Page

The Results page displays the calculated results of the objects in the GAP model.

Figure 7.7



Use the radio buttons at the top of the Results page to access the calculated results of the following objects:

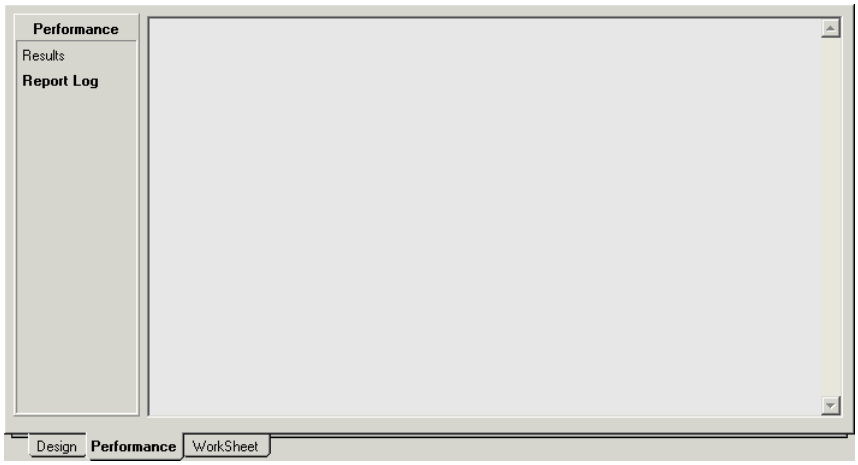
- Wells
- Separators
- Pipes

- Chokes
- Tanks

Report Log Page

The Report Log page displays the calculation logs.

Figure 7.8



7.2.3 Worksheet Tab

Refer to **Section 1.3.1 - Worksheet Tab** in the **HYSYS Operations Guide** for more information.

The Worksheet tab contains a summary of the information contained in the stream property view for all the streams attached to the operation.

Index