



## Jump Start: Aspen HYSYS<sup>®</sup> Dynamics V7.3

---

*A Brief Tutorial (and supplement to training and online documentation)*

*Glenn Dissinger, Product Director, Aspen Technology, Inc.*

*Julie Levine, Associate Product Marketing Professional, Aspen Technology, Inc.*

*Zachary Peers, Product Management Professional, Aspen Technology, Inc.*

## CONTENTS

<b>Introduction</b> .....	<b>2</b>
<b>Start with a Steady State Model</b> .....	<b>2</b>
<b>The Dynamics Assistant</b> .....	<b>3</b>
<b>Specify Pressure and Flow Parameters</b> .....	<b>4</b>
Pressure Node and Resistance Equation Operations .....	4
Boundary Stream Pressure Specifications.....	5
Dynamics Color Scheme .....	6
<b>Size the Equipment</b> .....	<b>7</b>
<b>Design a Control Strategy</b> .....	<b>8</b>
The Controller Library .....	8
The PID Controller.....	9
Setting a PID Controller .....	9
Face Plates.....	11
<b>Running Your Model</b> .....	<b>11</b>
<b>Strip Charts</b> .....	<b>12</b>
Generating a Strip Chart .....	12
Formatting a Strip Chart .....	13
<b>Transfer Functions</b> .....	<b>14</b>
<b>Event Scheduler</b> .....	<b>15</b>
Add a Sequence.....	15
Add an Event.....	16
Using Logical Expressions.....	16
Specify Event Actions.....	17
Activating Sequences.....	18
<b>Additional Resources</b> .....	<b>19</b>

## Introduction

Dynamic modeling of process simulations is vital to plant design and operation. Although there are many scenarios in which steady-state modeling is sufficient, accuracy can be significantly improved by accounting for a system's dynamic response to changes. Additionally, dynamic simulation is necessary to model a variety of scenarios including (but not limited to):

- Plant startup
- Plant shutdown
- Reactor runaway
- Pressure relief

Aspen HYSYS Dynamics allows users to easily convert steady state Aspen HYSYS simulations into powerful dynamic models. It runs entirely from within the regular Aspen HYSYS environment and uses the same input data and thermodynamic models, so users do not need to be concerned with inconsistencies or learning how to use a new interface.

This document serves as a simple "getting started guide" for using Aspen HYSYS Dynamics. We will take you through the most common progression of how to convert a steady state model to a dynamic one, and how to analyze it. This is not meant to be used as a stand-alone reference document. We recommend that a range of other resources be called upon to give the new user a comprehensive view of how to use dynamic modeling in Aspen HYSYS. These may include:

- AspenTech support website ([support.aspentech.com](http://support.aspentech.com)) – contains a wide range of knowledge base items and provides answers to frequently asked questions.
- AspenTech courseware available in on-line and in-person versions
- AspenTech business consultants

This document covers dynamic modeling in Aspen HYSYS. This guide assumes that the user has Aspen HYSYS V7.3 installed on their computer, but is also valid for some older versions of Aspen HYSYS.

## Start with a Steady State Model

The first step to dynamically simulating a process with Aspen HYSYS is to establish a steady state model. This model will contain all of the base inputs about the system: components, unit operations, thermodynamics, etc. The steady state model must be complete and have a converged solution.

This guide is focused on updating and running existing steady state models in Dynamics Mode. If you do not know how to build a steady state flow sheet in Aspen HYSYS, you should first consult reference material on this subject. We recommend the following:

- AspenTech support site
- Getting Started with Aspen HYSYS Computer Based Training module
- Aspen HYSYS training course

## The Dynamics Assistant

The Dynamics Assistant allows you to see what adjustments you need to make for your model to run dynamically and suggests changes. *You should use the assistant as a guide, not a cure-all.* To create the best possible models, it is important to use your own engineering judgment. It is strongly advised that the first time you convert a steady state model, you try to make all of the appropriate changes yourself before consulting the assistant.

To use the assistant, click the Dynamics Assistant button, next to the Dynamics Mode button, while you are still in Steady State Mode (Figure 1).



Figure 1: The Dynamics Assistant button

In the Dynamics Assistant pane, you will be given a list of parameters that you need to adjust and a “Make Changes” column. Notice that all of the items in this column are automatically checked. If you click the Make Changes button immediately, all of them will be made at once. To review the changes beforehand, double click any of the items on the list to redirect to the appropriate tab (Figure 2).

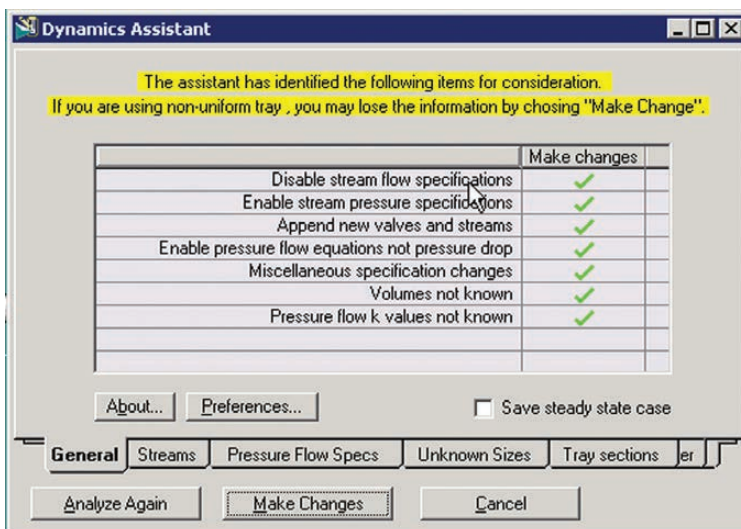


Figure 2: Double click any item in the list to redirect to the appropriate tab

On these tabs, you can see exactly what changes the Dynamics Assistant suggests. You can edit these changes directly in this window before clicking Make Changes, or you can make them in the PFD itself.

To make only certain changes with the Dynamics Assistant, be sure to click the boxes in the “Make Changes” column to uncheck them. No matter what tab in the assistant you are on, clicking the Make Changes button will change everything that is checked on the General tab.

## Specify Pressure and Flow Parameters

In Aspen HYSYS steady state models, pressure and flow parameters can be specified independently. For this reason, it is possible to “cheat” by making unrealistic specifications such as having no pressure drop across a unit. In HYSYS Dynamics, pressure and flow are related such that one cannot be calculated without the other. If you specify a zero pressure drop across a resistance unit in Dynamics, the flow rate through that unit will be zero (Figure 3).

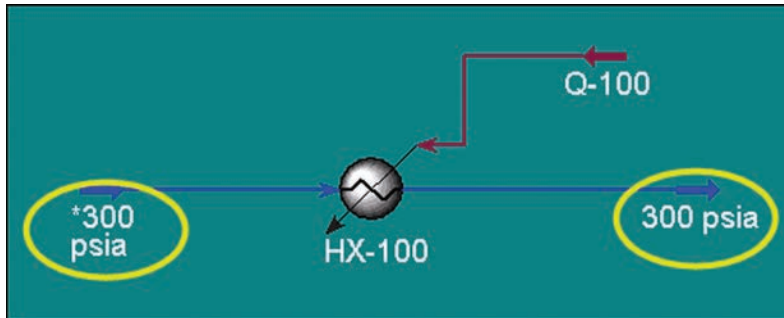


Figure 3: You must specify a pressure drop across resistance operations to run a dynamic simulation

### Pressure Node and Resistance Equation Operations

All unit operations can be classified as either pressure node operations or resistance equation operations:

- Pressure Node Operations: calculate a pressure drop based on the vapor holdup in the unit (vessels, etc.)
- Resistance Equation Operations: calculate a pressure drop based on a resistance equation (valves, pumps, compressors, etc.)

Your PFD must have a resistance operation in between every adjacent pressure node. Without this, the solver will assume that the pressures in the adjacent vessels are equal with no flow between them. Figure 4 illustrates this with a sample PFD. Without VLV-101, the two separators would have the same pressure with no flow between them.

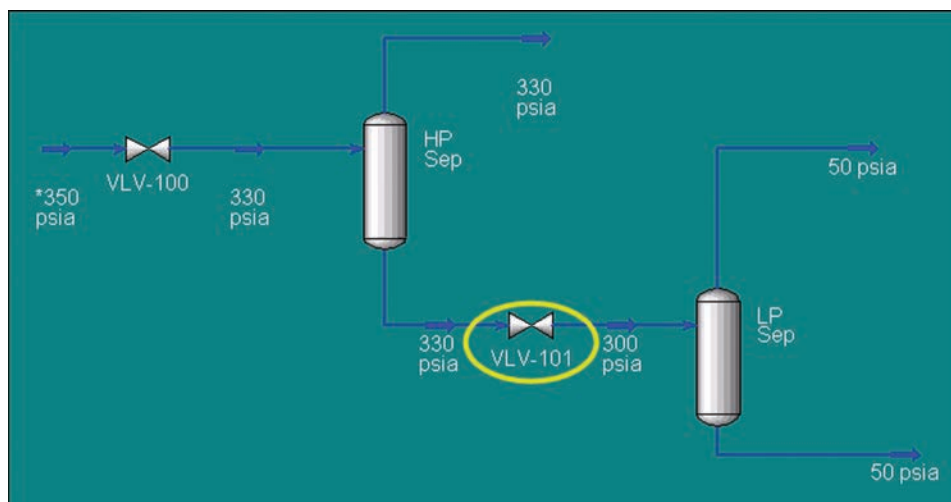


Figure 4: You must have a resistance operation between every pressure node, in this case a valve between two columns

Besides the fact that this modification is necessary to run a model in Dynamics Mode, it is good practice because the actual plant will have these features.

**Tips and Tricks:** To display all the stream pressures on your PFD, press Shift+P on your keyboard. To switch to temperatures, press Shift+T and to stream names, press Shift+N.

## Boundary Stream Pressure Specifications

Any boundary streams connected to a resistance operation need to have a pressure specification. Internal streams should not have pressure specifications, since their pressures should be determined by surrounding equipment. It is also recommended that you add a valve to all boundary streams. These can be used as flow controllers later when you develop a control system for your model.

To specify the pressure in a stream, right-click it and select View Properties to open its properties form (Figure 5). Then, adjust the pressure on the Dynamics tab. Take note of the dropdown that will appear on the right, and be sure that you are specifying with the correct units. Once you have specified the pressure, make sure that the Active box is checked (Figure 6).

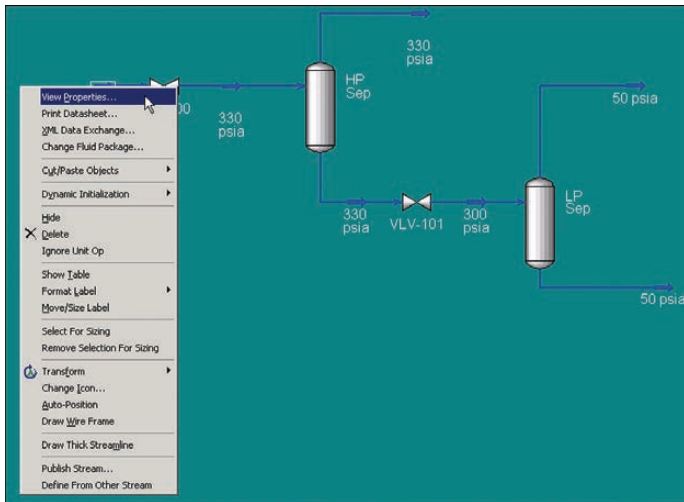


Figure 5: Open the Properties form for a stream

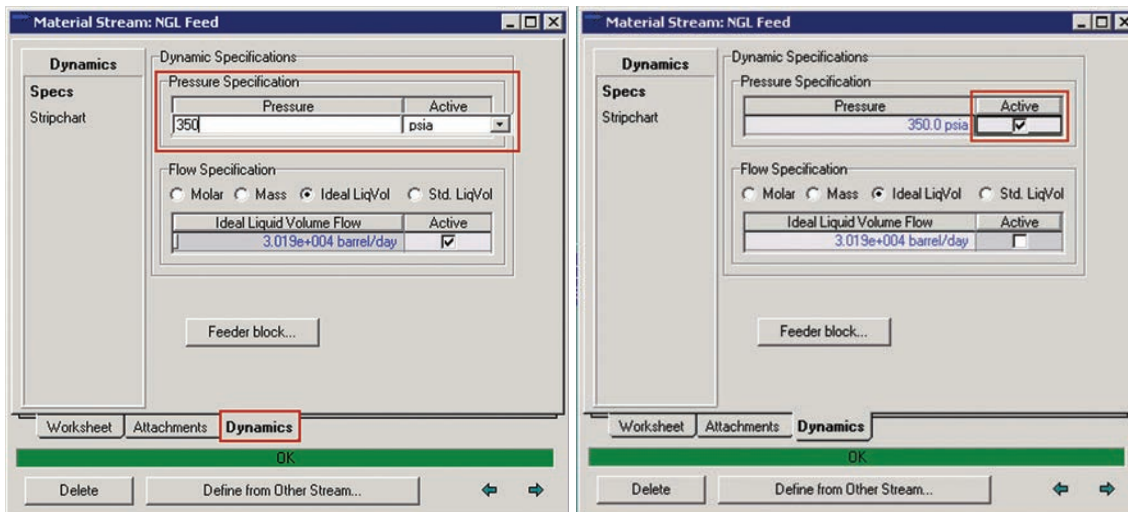


Figure 6: Specify a stream pressure on the Dynamics tab. Pay attention to the unit dropdown on the right!

## Dynamics Color Scheme

To make it easier to determine where you need to make pressure and flow specifications, you can change the color scheme of the PFD. In the upper right hand corner of the PFD, select the “Dynamic P/F Specs” color scheme from the dropdown (Figure 7). Table 1 shows what each color in this scheme indicates.

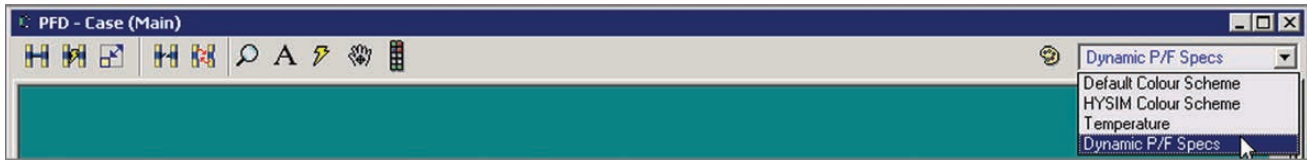


Figure 7: Select the Dynamics P/F Specs color scheme

Color	Meaning
Green	Pressure Specified
Yellow	Flow Specified
Red	Pressure and Flow Specified
Blue	No Specifications

Table 1: Decoding the Dynamics P/F Specs color scheme

Figure 8 shows a sample PFD with all required valves and pressure specifications. Notice that there are no red streams and that all of the boundary streams are green.

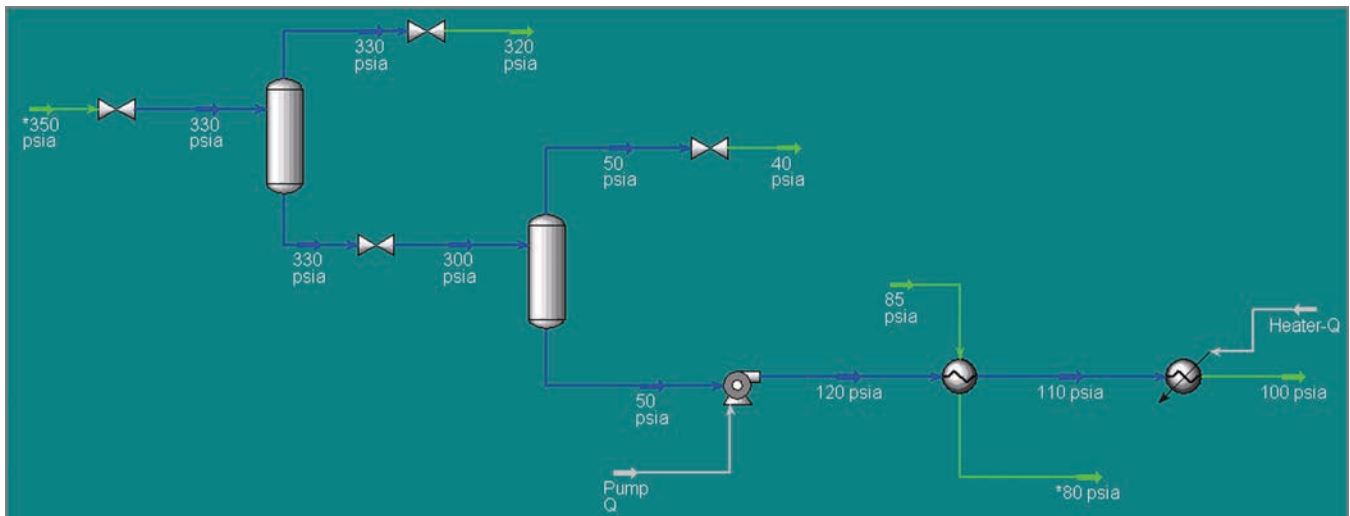


Figure 8: PFD with appropriate pressure and flow specifications for dynamic simulation

## Size the Equipment

In order to run dynamic simulations, it is important to have all of your equipment sized properly. The response of a vessel to fluctuations in its feed is highly dependent on the size of the vessel relative to the size of the disturbance. If your vessel is inappropriately large, the simulation will turn out an unrealistically dampened response. The reverse applies for a vessel that is too small.

There are two places where you can change the size of vessels: the Rating tab and the Dynamics tab on the vessel's property form (Figure 9).

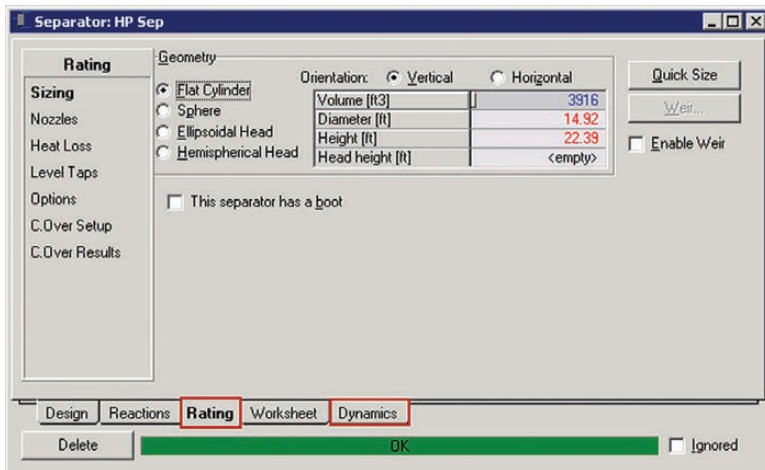


Figure 9: Size vessels from the Rating or Dynamics tab on their property forms

Additionally, you can use the Dynamics Assistant to auto-size your equipment. Do so by changing the residence time on the assistant to determine an appropriate volume (Figure 10).

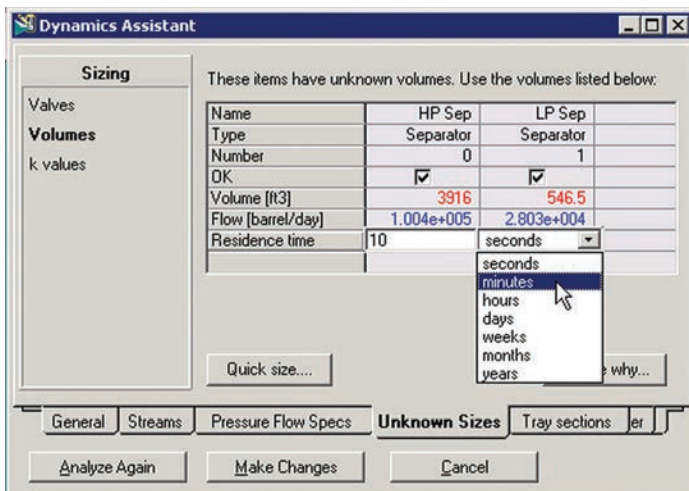


Figure 10: Set the residence time to determine vessel volume



**Note:** For simple heat exchangers, there is a default vessel size of 0.1m<sup>3</sup>. If you use the Dynamics Assistant, you will not be instructed to resize this, even though it is most likely not appropriate.


## Design a Control Strategy

Without a good control scheme, there is no way for a process to cope with disturbances. If unmitigated, even small fluctuations can throw an entire system out of sorts. As such, it is important to establish a control scheme before you start running your simulation dynamically.

### The Controller Library

Aspen HYSYS provides a variety of controllers and logical operations that you can use for your control scheme. To access these, you must open the appropriate palettes. First, open the Object Palette by either clicking the Object Palette button or selecting it from the Flow sheet menu in the toolbar.

To open the Controller Palette, click the Control Ops icon  on the lower left.

To open the Boolean Palette, click the Boolean Ops icon  on the lower right. Figure 11 shows the location of these buttons and the resulting palettes.

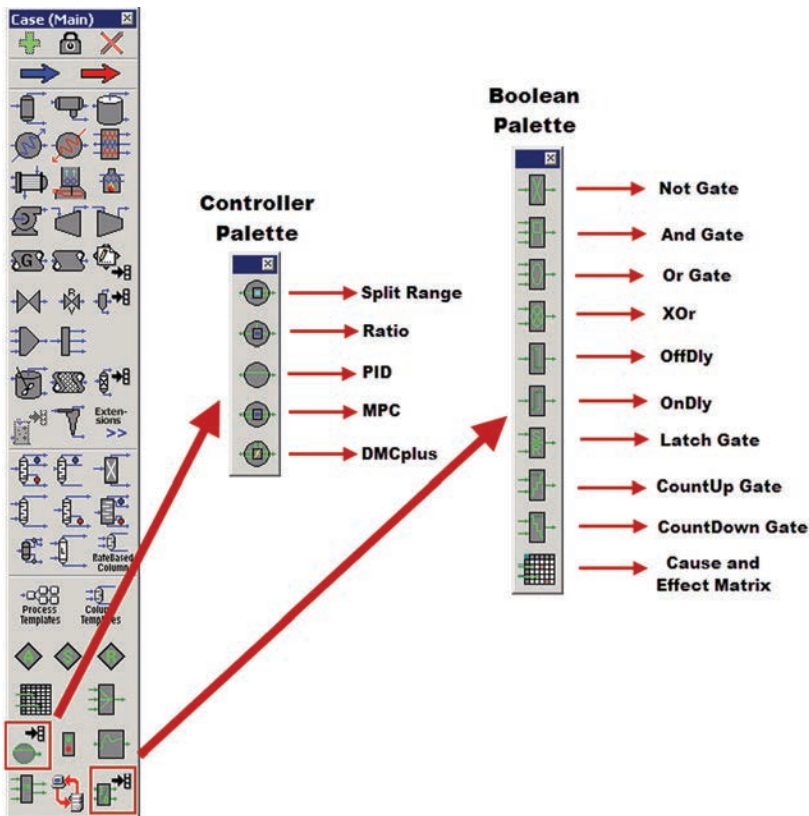


Figure 11: The Control Ops and Boolean Ops Palettes and their constituents

### The PID Controller

For most control schemes, you will primarily use proportional-integral-derivative, or PID, controllers. As the name suggests, these allow you to employ proportional, integral, and derivative control. The basic PID equation is:

$$OP(t) = K_c E(t) + \frac{K_c}{\tau_i} \int E(t) dt + K_c \tau_d \frac{dE(t)}{dt}$$

$OP(t)$  is the controller output as a function of time and  $E(t)$  is the error between the set point and process variable. The controller can be tuned by adjusting the controller gain,  $K_c$ , the integral reset,  $\tau_i$ , and the derivative gain,  $\tau_d$ . Table 2 shows our suggestions for initial PID tuning parameters. Note that  $\tau_i$  appears in the denominator of the PID equation; thus, increasing the integral reset will actually decrease the impact of the integral gain on the controller response.

System	$K_c$	$\tau_i$ (minutes)	$\tau_d$ (minutes)
Flow	0.1	0.2	0
Level	2	10	0
Pressure	2	2	0
Temperature	1	20	0

Table 2: PID tuning recommendations

### Setting a PID Controller

To employ a PID controller, you must instantiate one from the controller palette and then right-click and select View Properties. On the Connections tab, select the Process Variable and Output Target Object (Figure 12). The process variable is the variable value that the controller is trying to regulate. The output target object (commonly referred to as the manipulated variable), is the parameter that the controller will change to adjust the process variable to the set point.

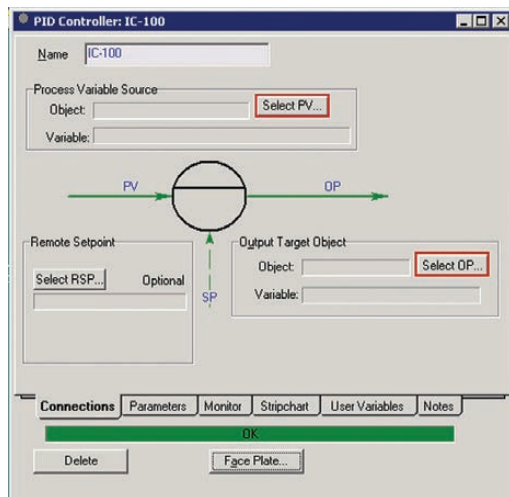


Figure 12: Select a process variable and output target object for a PID controller

Next, click the Parameters tab. Under Operational Parameters, set the action. If you want the output target object to have a reverse response to changes in the process variable, then select "Reverse." For example, consider Figure 13, where your process variable is the liquid level in a tank, and your output target object is valve A, which controls the inlet. When the liquid level falls you want the valve to open so the inlet flow will increase and raise the tank level. This is a reverse

response. If your output target object is valve B, which controls the outlet flow, you want a direct response. That way, when the liquid level falls the valve will close, reducing the outlet flow rate and raising the tank level.

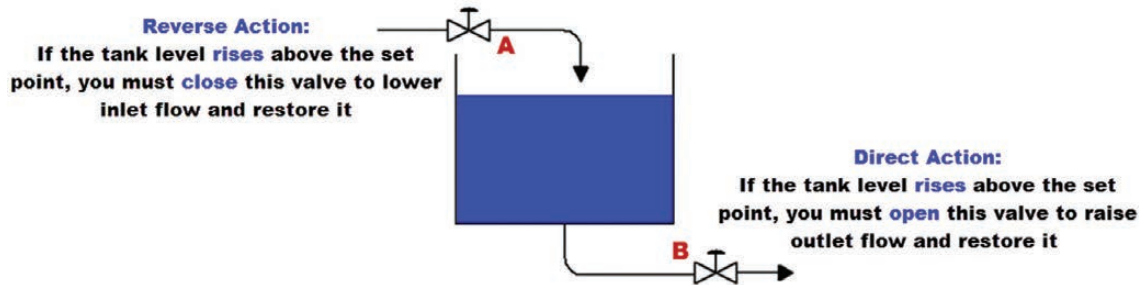


Figure 13: Direct vs. reverse controller action

Next, specify the set point and the PV Minimum and Maximum values. Finally, set the tuning parameters (see the recommendations in Table 2). Figure 14 shows the Parameters tab for a PID controller and highlights all of the inputs you should be sure to specify. Finally, you can activate the PID controller by switching the Mode to “Auto”.

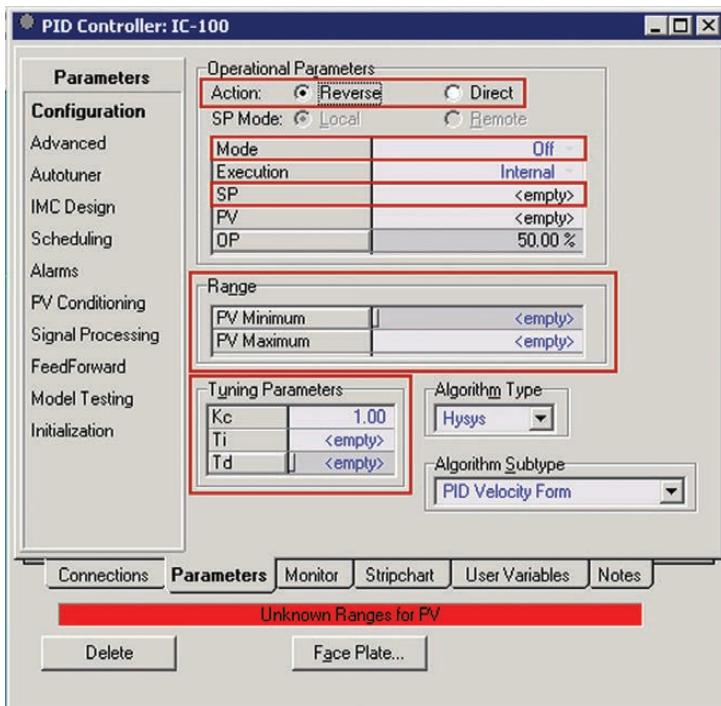


Figure 14: The Parameters tab for a PID controller

## Face Plates

For every controller, you can view a face plate. Either select the face plate button from the properties form or right-click on the controller and select “Face Plate...” The top bar on the face plate shows the process variable value (blue) relative to the set point (red arrow). The bottom bar shows the controller output (blue). It is possible to manipulate the set point directly on the face plate by moving the red arrow (Figure 15).



Figure 15: Controller face plate: move the red arrow to adjust the set point, and see how the process variable and controller output respond while the simulation is running

## Running Your Model


Now that you have fixed your pressure and flow specifications, sized all of your equipment, and established a control strategy, you are ready to run your model in Dynamics Mode. Use the Dynamics Assistant to make sure that you are not missing any important specifications and then click the Dynamics Mode button  in the toolbar (Figure 16).



Figure 16: The Dynamics Mode button

## Strip Charts

Strip charts are a way to observe the dynamic fluctuations of your model graphically. They can be produced for any stream, block, or controller.

### Generating a Strip Chart

There are several ways to produce strip charts. The first is to open the properties form for a stream or unit operation and navigate to the Dynamics tab. Open the Strip Chart page to see several default variable sets. Select one from the Variable Set dropdown and click the Create Strip Chart button (Figure 17). For controllers, there is a Strip Chart tab.

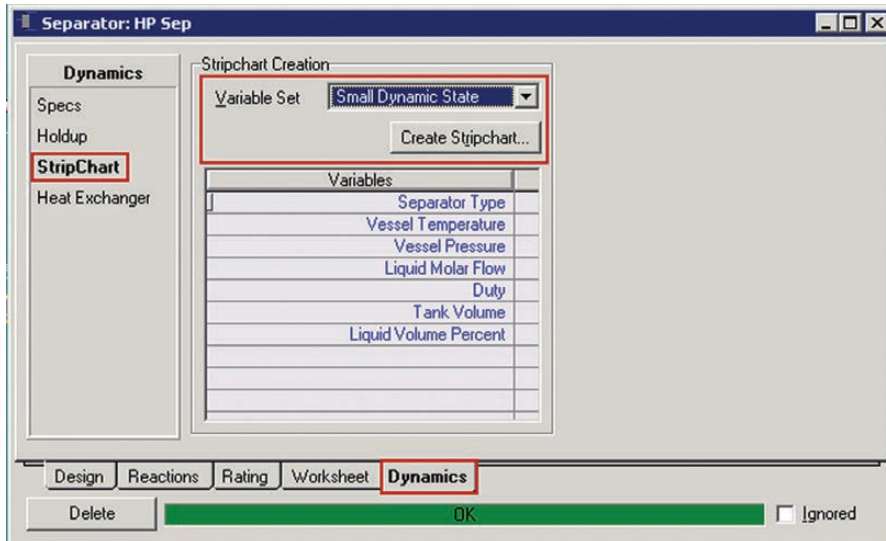


Figure 17: Generate a strip chart for a unit operation using a default variable set

You can also generate a strip chart using the DataBook. Open the DataBook from the Tools menu in the toolbar, or press Ctrl+D on the keyboard. Next, navigate to the Strip Charts tab. Click the Add button to create a new DataLogger and name it in the Logger Name section. Notice that the Object and Variable columns will automatically be filled with possible variables for your strip chart. In the Active column on the right, you can check off which of these you want to plot (Figure 18).

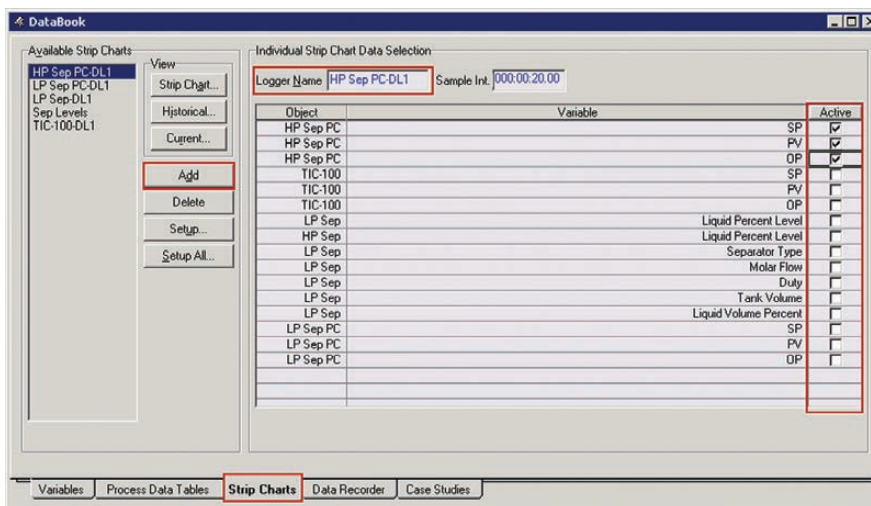


Figure 18: Create a new DataLogger, rename it, and select active variables for your strip chart

Next, click the Setup button and change the logger size and sample interval (Figure 19). With the standard setting you will only log for 100 minutes of plant time. We recommend a logger size with an order of magnitude of three to start.

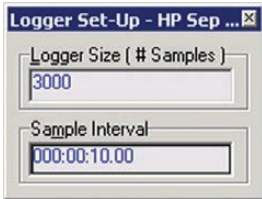


Figure 19: Change the logger size and sample interval

To generate the chart, click the Strip Chart button on the DataBook panel in the View Section. If you are in Integrator Active mode (the green light in the toolbar is selected), the chart will automatically start generating information.

### Formatting a Strip Chart

To format a strip chart, right-click on the plot area and select “Graph Control...” (Figure 20).

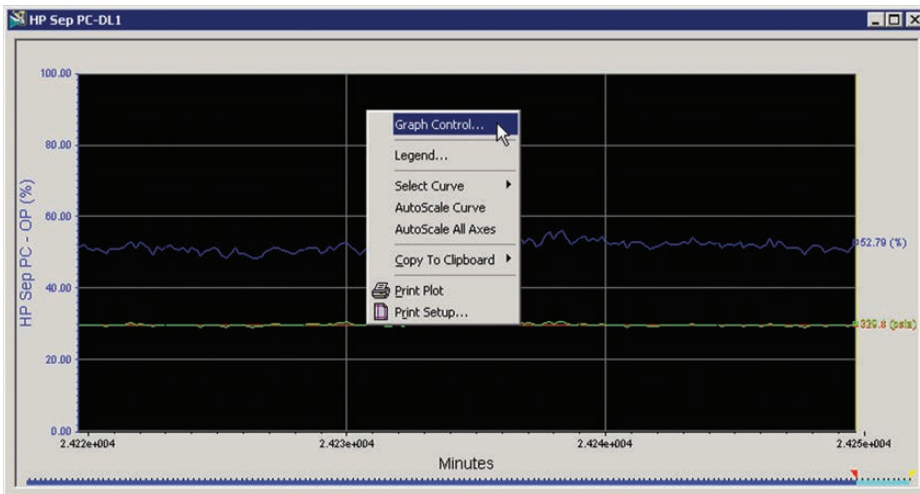


Figure 20: Open graph control for a strip chart

This will open the Strip Chart Configuration pane (Figure 21). From this pane, you can adjust how each variable is presented, the axes, and other display parameters for this particular chart.

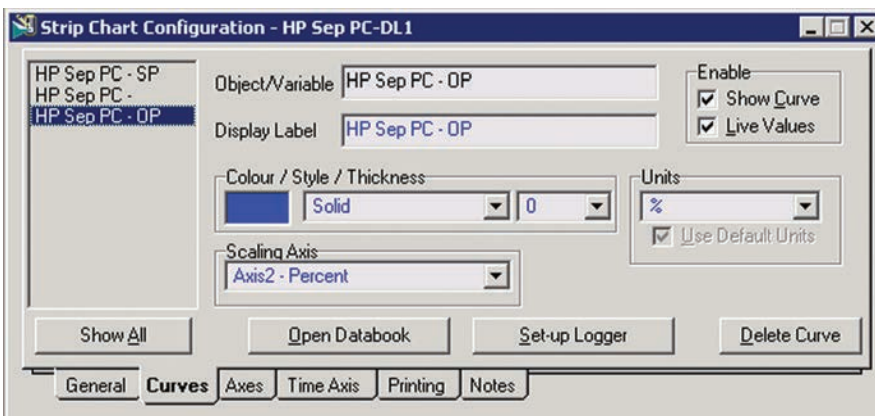



Figure 21: Strip Chart Configuration pane

## Transfer Functions

One of the useful features of dynamic modeling in Aspen HYSYS is that it allows you to observe how a simulation will respond to disturbances. You can model these disturbances using transfer functions or by adding random noise to any process variable. To implement a transfer function on your PFD, select the Transfer Function icon  on the lower right corner of the object palette (Figure 22).

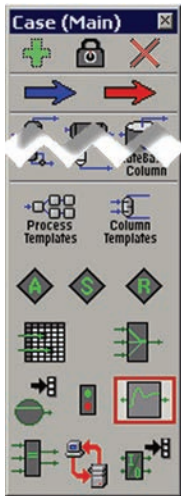


Figure 22: The Transfer Function icon

Once you have instantiated a transfer function, open its property form. From the Connections page, you can select the target stream, unit operation, or controller that you want the transfer function to effect (Figure 23).

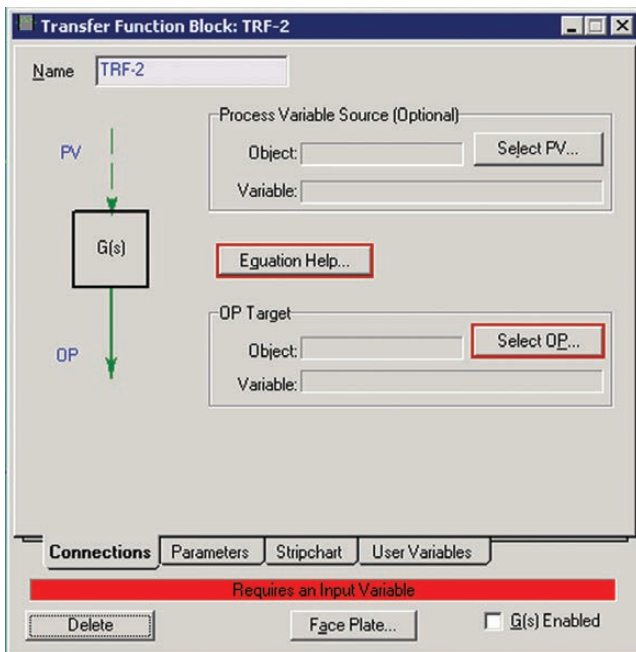


Figure 23: Select the target for a transfer function

On the Parameters tab you can fill out the remaining inputs for your transfer function. You must fill in the ranges for the PV and OP in the Ranges section before you can fill in the Operational Parameters (Figure 24).

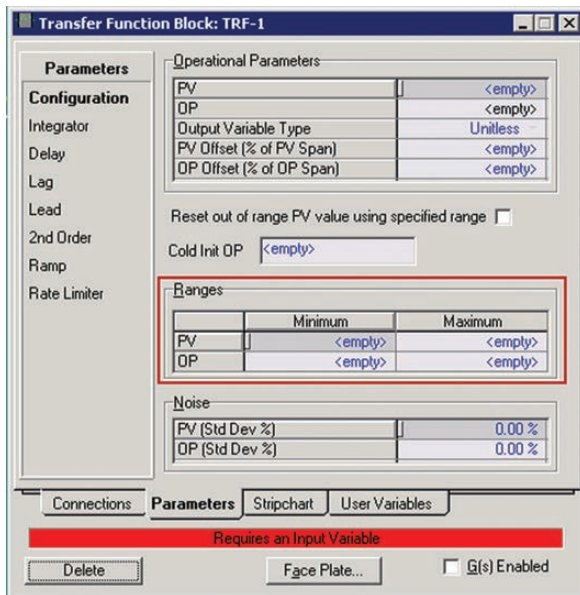


Figure 24: Fill out the parameters for a transfer function. Remember to fill in the Ranges first!

In the Noise section, you can choose to add noise to your signal, to see how your control system responds to a more realistic input.

## Event Scheduler

The event scheduler allows users to have Aspen HYSYS automatically perform events. These can be triggered by elapsed time, logical operations, or the value of a variable stabilizing. The event scheduler is especially helpful in evaluating emergency scenarios in a plant, as well as startup and shutdown sequences. It can additionally be used as a training tool to see how new operators respond to various disturbances.

### Add a Sequence

To schedule events, open the Event Scheduler from the Simulation menu in the toolbar (or press Ctrl+E on the keyboard). Next, click the Add button in the Schedule Options section. This will open the Schedule Sequences group. To add a sequence of events, click the Add button on the right. This will add a sequence to the matrix. Here, you can give the sequence a more descriptive name and determine if you want to act in one shot or continuously. Next, select the sequence and click the View button (Figure 25).



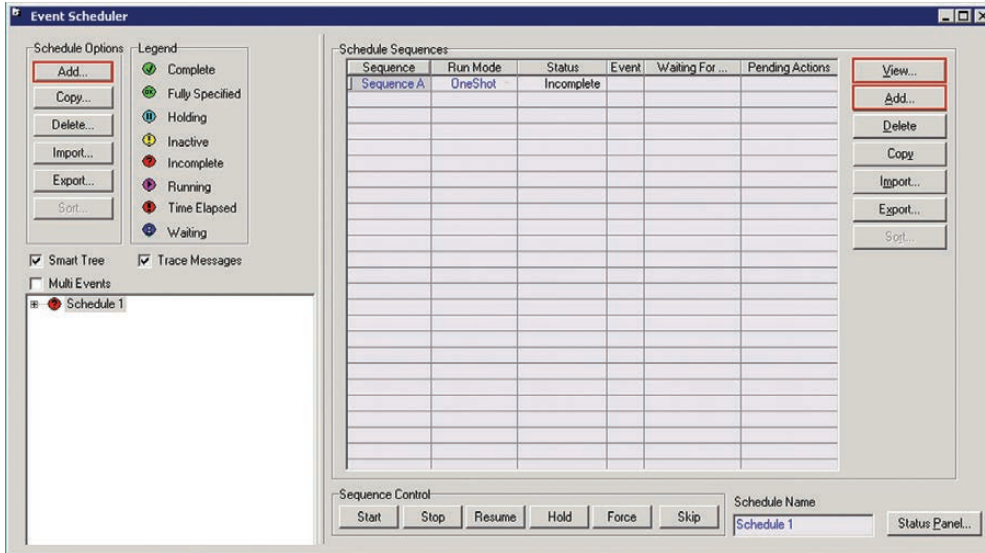


Figure 25: Add a schedule and then a sequence and click View to see the individual events

### Add an Event

In the Sequence pane that appears, click the Add button to add an event. You can add as many events as you'd like to a sequence. Next specify the conditions and actions associated with each event in the matrix. To do so for a given event, double click on it (Figure 26). On the pane that follows, you can specify how you want the event to be triggered on the Condition tab.

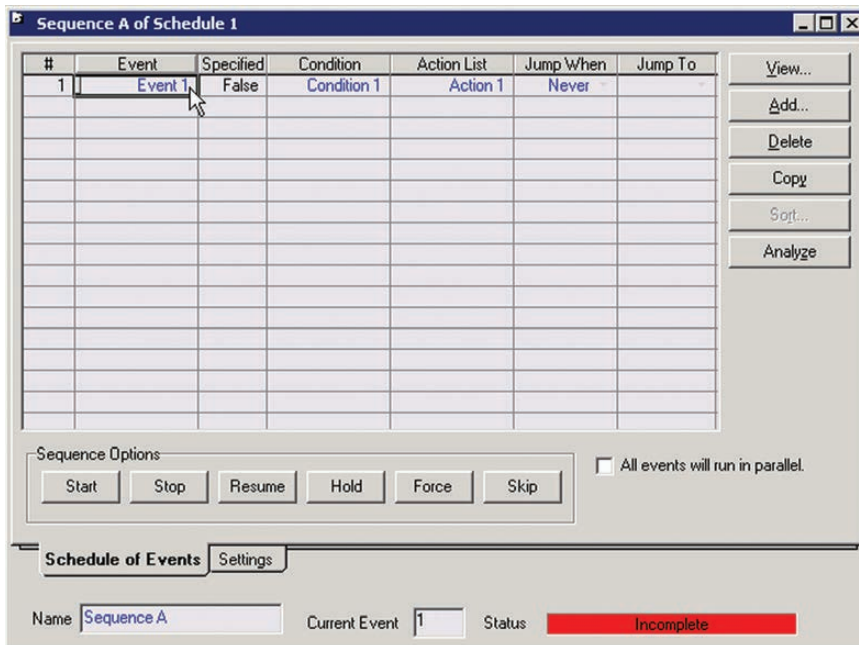


Figure 26: Edit a specific event

### Using Logical Expressions

To use logical expressions, you must first define the variables in the Variables for Logic Expression section. Click Add Variable and select any that you will reference. This will create tags which you can use when writing the expression. For help with syntax, click the Expression Help button in the Logical 'IF' Expression section. Enter the expression in the box.

Figure 27 shows a sample expression, which states that the event is triggered when the volumetric flow of the stream NGL Feed reaches 90,000 barrels/day.

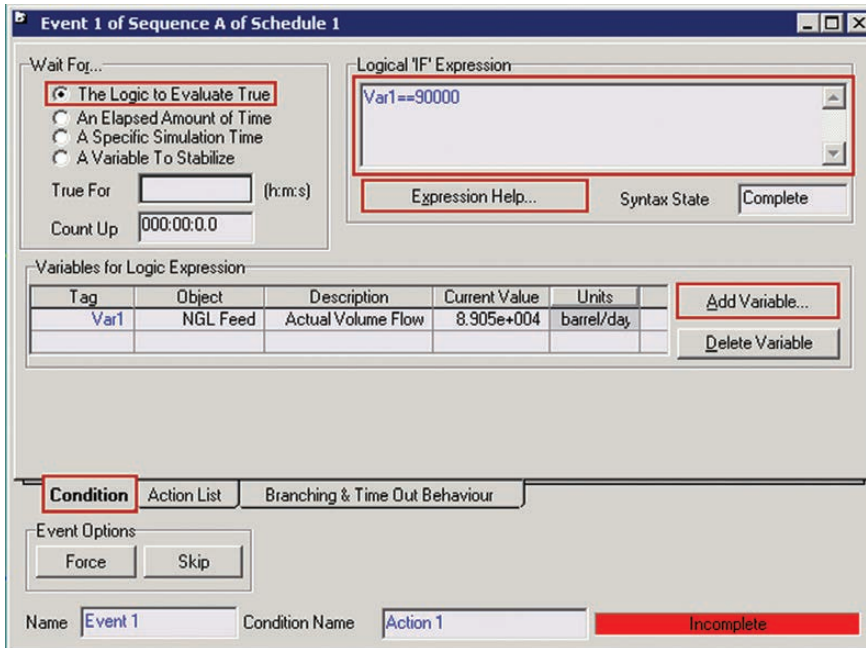


Figure 27: Trigger an event when the volumetric flow rate of NGL Feed is 90,000 barrels/day

### Specify Event Actions

Once you have specified the trigger for an event, you must specify what action(s) will occur on the Action List tab. You can add multiple actions for each event by clicking the Add button in the List Of Actions For This Event section. For each separate action, fill out the Individual Action Specification (Figure 28).

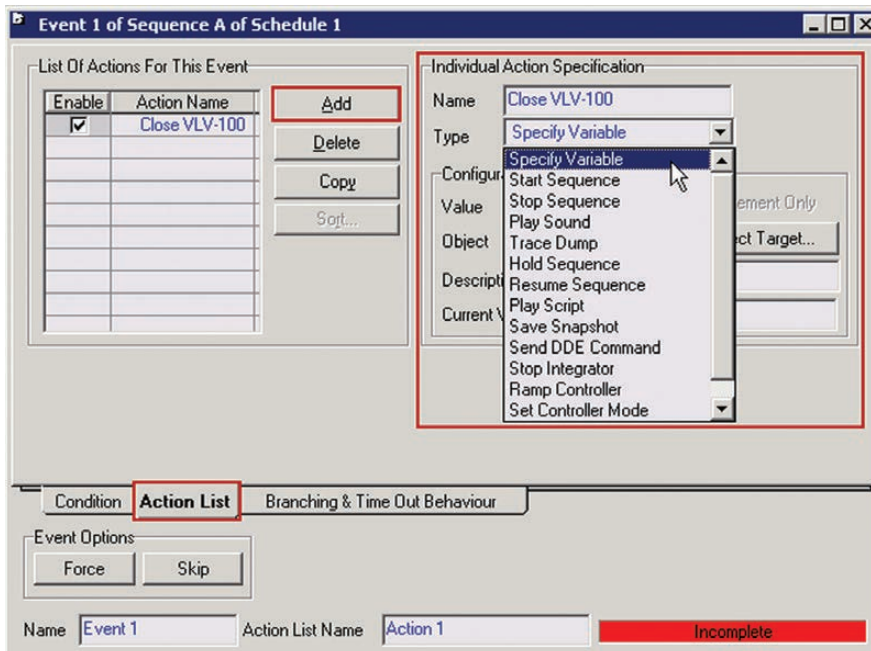


Figure 28: Add actions and configure specifications

**Note:** If you have a control scheme in place (which you should!) specifying a value will cause it to change, but it will not remain at that value because your controllers will continue to act on it.

### Activating Sequences

To run a sequence while the solver is active, click the Start button in the Sequence control group on the Event Scheduler (Figure 29). When all of the events have terminated, the status will be Complete. You can also Stop, Resume, Hold, Force, and Skip a sequence by clicking the corresponding buttons in the Sequence Control group.

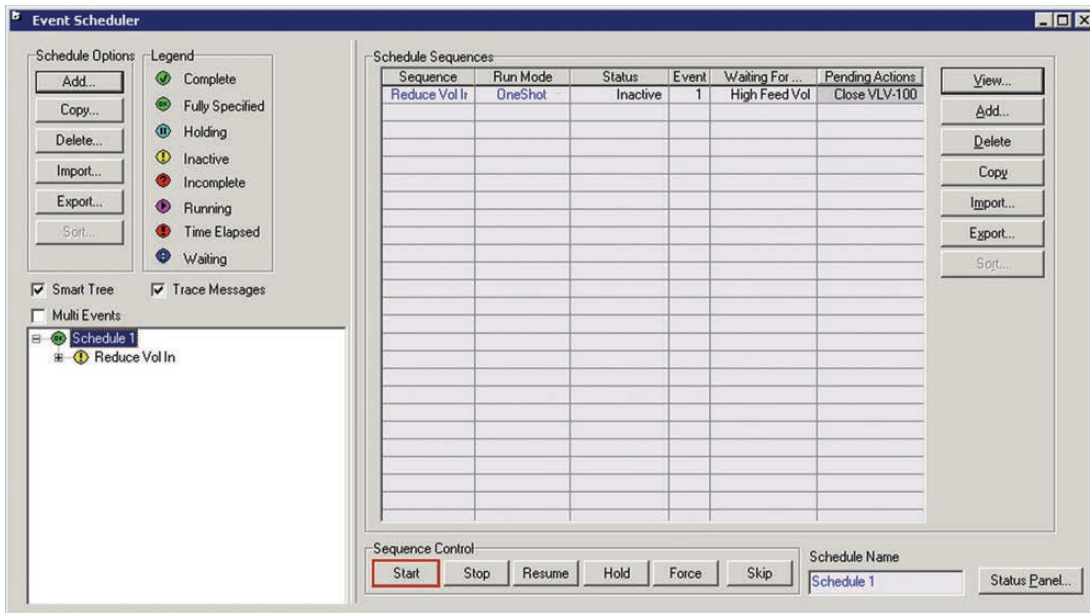


Figure 29: Start a given sequence

## Additional Resources

For further information on Aspen HYSYS please consult:

**Public Website:**

<http://www.aspentech.com/products/aspens-hysys.aspx>

**Support Website:**

The support website provides an extensive and growing knowledge base as well as Computer Based Training (CBT) Courses

[http://support.aspentech.com/webteamasp/My/FrameDef.asp?/webteamasp/My/product.asp?id1=2674&id2=""&id3=all](http://support.aspentech.com/webteamasp/My/FrameDef.asp?/webteamasp/My/product.asp?id1=2674&id2=)

**Solving Design and Operational Challenges Using Dynamic Simulation On-Demand Webinar:**

<https://aspentechevents.webex.com/aspentechevents/lsrphp?AT=pb&SP=EC&rID=56402112&rKey=8846d75ac81dabe6>

## ||||||| About AspenTech

AspenTech is a leading supplier of software that optimizes process manufacturing—for energy, chemicals, pharmaceuticals, engineering and construction, and other industries that manufacture and produce products from a chemical process. With integrated aspenONE® solutions, process manufacturers can implement best practices for optimizing their engineering, manufacturing, and supply chain operations. As a result, AspenTech customers are better able to increase capacity, improve margins, reduce costs, and become more energy efficient. To see how the world's leading process manufacturers rely on AspenTech to achieve their operational excellence goals, visit [www.aspentech.com](http://www.aspentech.com).



### Worldwide Headquarters

Aspen Technology, Inc.  
20 Crosby Drive  
Bedford, MA 01730  
United States

phone: +1-781-221-6400  
fax: +1-781-221-6410  
[info@aspentech.com](mailto:info@aspentech.com)

### Regional Headquarters

**Houston, TX | USA**  
phone: +1-281-584-1000

**São Paulo | Brazil**  
phone: +55-11-3443-6261

**Reading | United Kingdom**  
phone: +44-(0)-1189-226400

**Singapore | Republic of Singapore**  
phone: +65-6395-3900

**Manama | Bahrain**  
phone: +973-17-50-3000

For a complete list of offices, please visit  
[www.aspentech.com/locations](http://www.aspentech.com/locations)