

See discussions, stats, and author profiles for this publication at: <https://www.researchgate.net/publication/333783728>

# Aspen HYSYS (DYNAMICS) Training Course

Book · June 2019

---

CITATIONS

0

---

READS

13,566

5 authors, including:



[Sameh Tawfeek](#)

Arab Academy for Science, Technology, and Maritime Transport

117 PUBLICATIONS 21 CITATIONS

SEE PROFILE



# Aspen HYSYS (DYNAMICS) Training Course

By :-



**Dr \ Sameh Tawfik Abd El-fattah Mahmoud**

Assistant Professor of Cryogenic & Gas liquefaction

Faculty of engineering – Alexandria University

Email: sameh\_tawfeek@hotmail.com  
sameh\_tawfeek@alexu.edu.eg  
Mobile: 01001808616 - 01274422545

# Table of Contents

Introduction .....	1
Example Files .....	1
Start with a Steady State Model .....	2
The Dynamic Assistant .....	2
Specify Pressure and Flow Parameters .....	4
Pressure Node and Resistance Equation Operations .....	4
Boundary Stream Pressure Specifications .....	5
Dynamic Color Scheme .....	6
Size the Equipment .....	7
Design a Control Strategy .....	8
Controllers on the Model Palette .....	8
The PID Controller .....	8
Setting a PID Controller .....	9
Face Plates .....	10
Running Your Model .....	12
Strip charts .....	12
Generating a Strip Chart .....	12
Formatting a Strip Chart .....	14
Transfer Functions .....	15
Event Scheduler .....	17
Add a Sequence .....	17
Add an Event .....	18
Using Logical Expressions .....	18
Specify Event Actions .....	19
Activating Sequences .....	20
Additional Resources .....	21

## Introduction

Dynamic process modeling is vital to plant design and operation. Although many times 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 Dynamic allows you to easily convert steady state Aspen HYSYS simulations into powerful dynamic models. It runs entirely from within the familiar Aspen HYSYS environment and uses the same input data and thermodynamic models.

This document serves as a simple “getting started guide” for using Aspen HYSYS Dynamic. 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 you use a range of other resources including:

- AspenTech support website ([support.aspentech.com](http://support.aspentech.com))
- AspenTech courseware available in on-line and in-person versions
- AspenTech knowledgebase items, training, and sample models available in aspenONE® Exchange
- AspenTech business consultants

This guide assumes that the user has Aspen HYSYS V8.0 or higher installed on their computer.

## Example Files

There are two example files used in this tutorial, HYSYS Dynamic Jump Start Case 1 and HYSYS Dynamic Jump Start Case 2. Case 1 is a steady state model and Case 2 is the same model after it has been converted into a dynamic one. This guide shows an overview of how to transform Case 1 into Case 2, but it does not show every individual change. To get more familiar with upgrading steady state models to dynamic ones, we recommend you try to accomplish this on your own. Versions of these files for Aspen HYSYS V7.3 are also featured in the On-Demand Webinar Solving Design and Operational Challenges Using Dynamic Simulation (Additional Resources, page 21).

Both example files are available for free as an instant download through aspenONE Exchange. To access aspenONE Exchange in Aspen HYSYS V8.0, click the Exchange button in the Get Started tab on the ribbon (Figure 1). In Aspen HYSYS V8.2 and higher, aspenONE Exchange automatically launches when you open the program.



Figure 1: Open aspenONE Exchange

Type “HYSYS Dynamic Jump Start” into the search bar and click on the appropriate flow sheets in the search results (Figure 2). You can also find other example files, knowledgebase tips, training modules, and more using similar queries. To learn more about how to use aspenONE Exchange, see the associated Jump Start in the Additional Resources section.

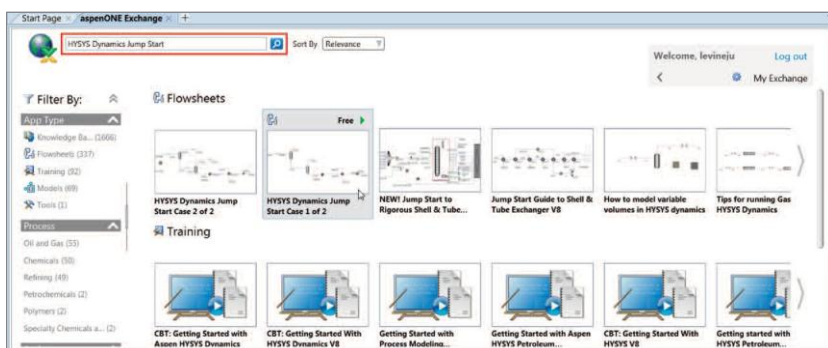


Figure 2: Search for example files

Note, the workflow demonstrated here is not specific to these example files and can be applied to any case.

## 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 for the system: components, unit operations, thermodynamic, 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 Dynamic Mode. If you do not know how to build a steady state simulation in Aspen HYSYS, see the associated Jump Start in the Additional Resources section (Page 21).

## The Dynamic Assistant

The Dynamic 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 Dynamic Assistant button (located in the Dynamic Simulation group on the Dynamic tab) while you are still in Steady State Mode (Figure 3).

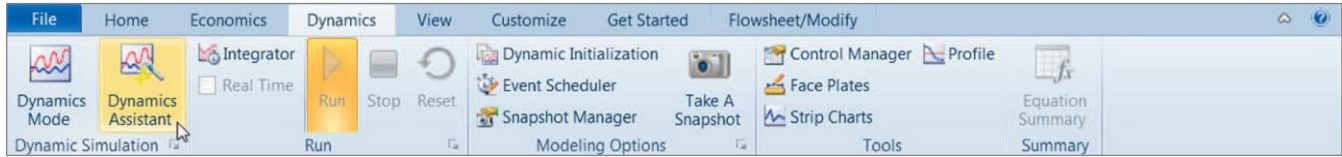


Figure 3: The Dynamic Assistant button

In the Dynamic Assistant window, you will be given a list of parameters to adjust. Notice that all of the items are automatically checked in the “Make changes” column. If you click the Make Changes button, all checked changes will be made at once no matter what tab you are on. To make only certain changes, uncheck the ones you don’t want (Figure 4).

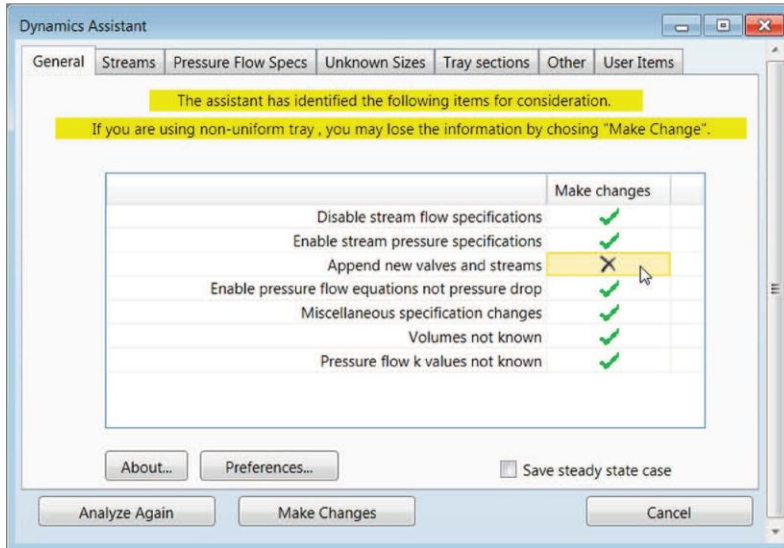


Figure 4: Click on the boxes in the “Make changes” column to uncheck desired items

To review individual changes beforehand, double click any of the items on the list to redirect to the appropriate tab (Figure 5).

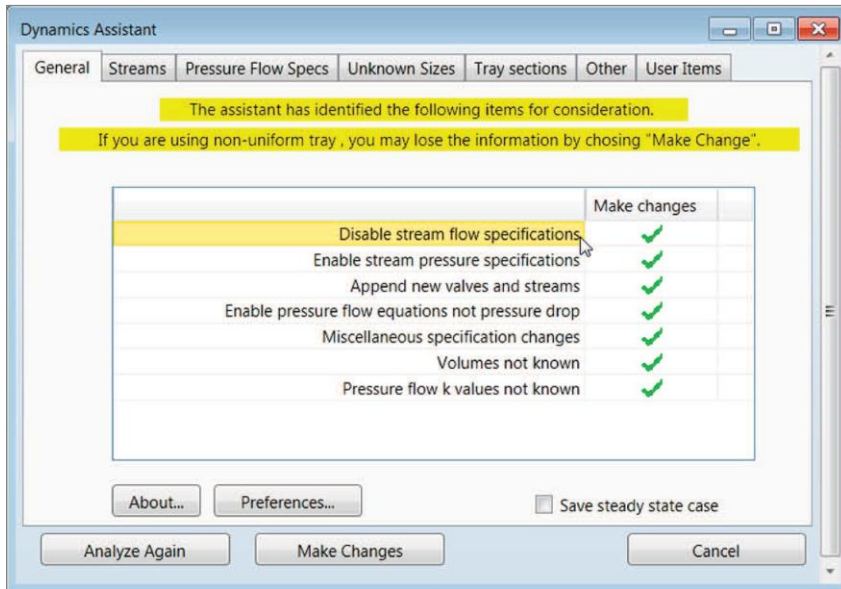


Figure 5: Double click any item in the list to redirect to the appropriate tab

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

## 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 Dynamic, pressure and flow are related such that one cannot be calculated without the other. If you specify zero pressure drop across a resistance unit in Dynamic, the flow rate through that unit will be zero as well (Figure 6).

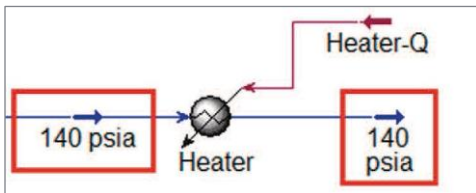


Figure 6: You must specify a pressure drop across resistance operations to run a dynamic simulation

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

Your flowsheet 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 7 shows an example of this from Case 1. Without VLV-101, the two columns have the same pressure with no flow between them.



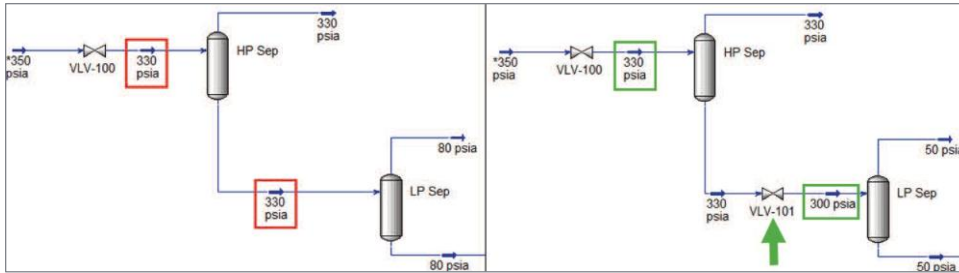


Figure 7: You must have a resistance operation between every pressure node, in this case a valve between two columns

Besides the fact that this modification is necessary to run a model in Dynamic Mode, it is good practice because real plants have these features.

**Tips and Tricks:** To display all the stream pressures on your flowsheet, select “Pressure” from the list in the Stream Label group on the Flow sheet/Modify contextual tab on the ribbon (which appears when the flowsheet is selected).

## 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 (Design a Control Strategy, page 8).

To specify the pressure in a stream, double click on it to open its properties form. Adjust the pressure on the Dynamic tab. Take note of the unit dropdown that appears on the right when you type in a new number. Once you have specified the pressure, make sure that the Active box is checked (Figure 8). Note: Because the calculations are interdependent you must specify either pressure or flow, not both.

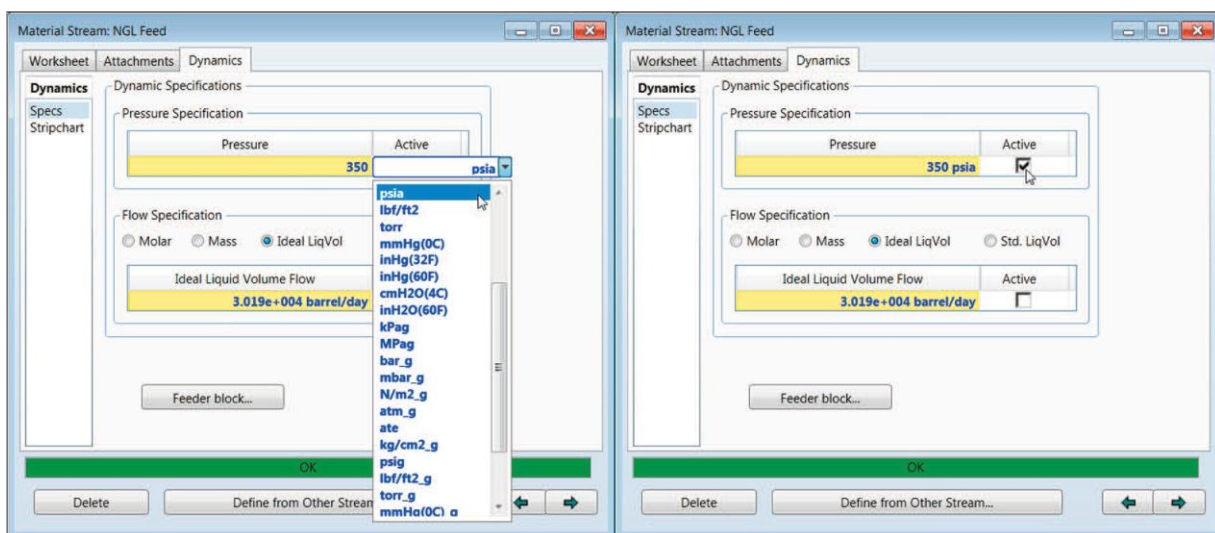


Figure 8: Specify a stream pressure on the Dynamic tab



## Dynamic 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 flowsheet. In the Display Options group on the Flowsheet/Modify tab, select the “Dynamic P/F Specs” color scheme from the dropdown (Figure 9). Table 1 shows what each color in this scheme indicates.

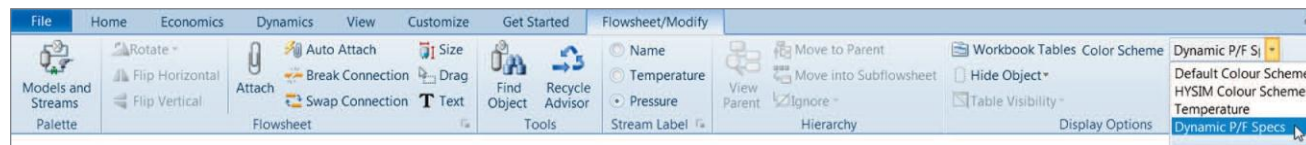


Figure 9: Select the Dynamic P/F Specs color scheme

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

Table 1: The Dynamic P/F Specs color scheme

Figure 10 shows Case 1 after all required valves and pressure specifications have been made. Notice that there are no red streams and that the boundary streams are green. The stream entering Heat Exchanger is yellow because it is a utility that is flow specified.

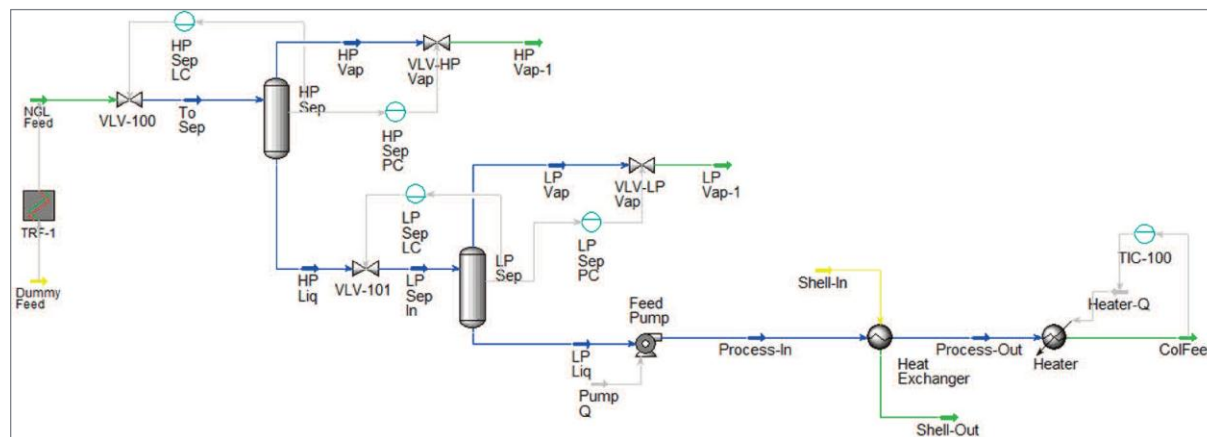
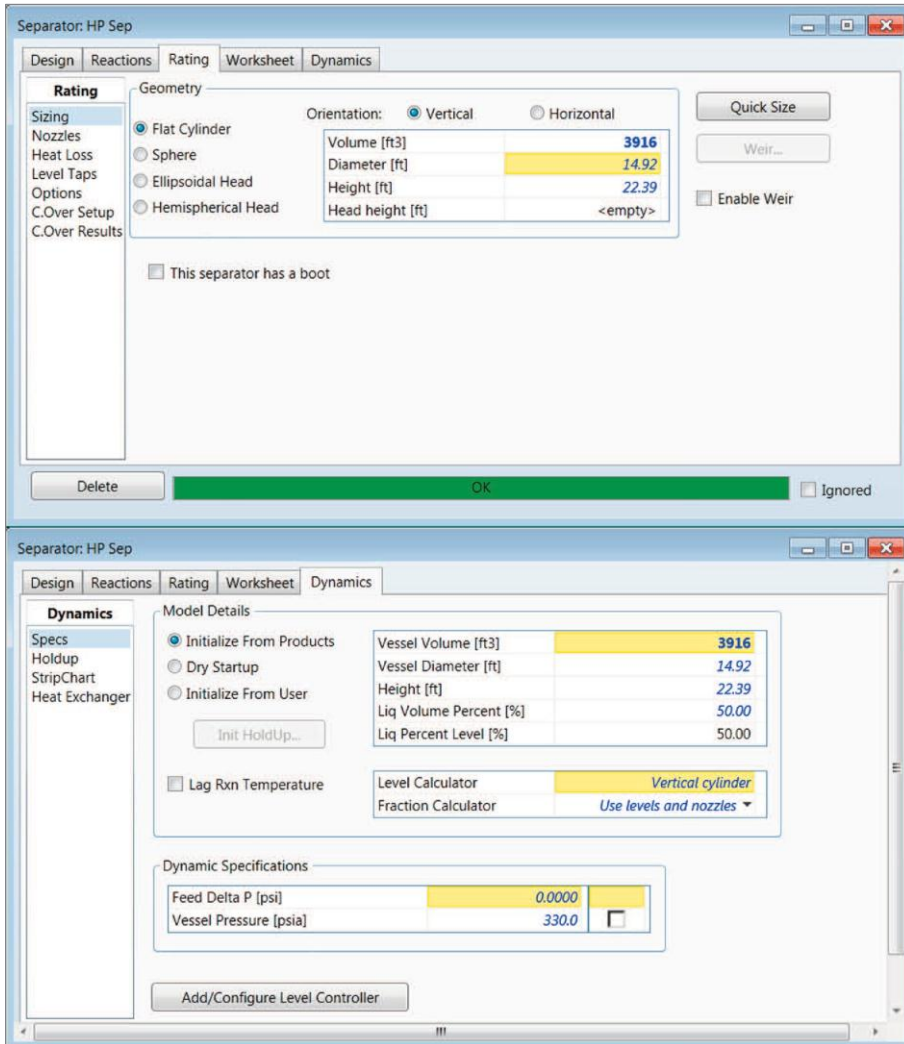


Figure 10: Flowsheet with appropriate pressure and flow specifications for dynamic simulation

## 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 Dynamic tab on the vessel's property form (Figure 11).



Separator: HP Sep

Design Reactions Rating Worksheet Dynamics

**Rating**

Geometry

Orientation: ☒ Vertical ☐ Horizontal

☒ Flat Cylinder ☐ Sphere ☐ Ellipsoidal Head ☐ Hemispherical Head

Volume [ft3]	3916
Diameter [ft]	14.92
Height [ft]	22.39
Head height [ft]	<empty>

☐ This separator has a boot

Buttons: Quick Size, Weir..., Enable Weir, Delete, OK, Ignored

---

Separator: HP Sep

Design Reactions Rating Worksheet Dynamics

**Dynamics**

Model Details

☒ Initialize From Products ☐ Dry Startup ☐ Initialize From User

Buttons: Init HoldUp...

Vessel Volume [ft3]	3916
Vessel Diameter [ft]	14.92
Height [ft]	22.39
Liq Volume Percent [%]	50.00
Liq Percent Level [%]	50.00

☐ Lag Rxn Temperature

Level Calculator: Vertical cylinder

Fraction Calculator: Use levels and nozzles

Dynamic Specifications

Feed Delta P [psi]	0.0000
Vessel Pressure [psia]	330.0

Buttons: Add/Configure Level Controller

Figure 11: Size a vessel from the Rating or Dynamic tab on its property form

Additionally, you can use the Dynamic Assistant to auto-size your equipment. On the unknown sizes tab, select volume on the left. Change the residence times for the vessels you want to size and click the Quick size... button (Figure 12). New sizes will automatically be propagated throughout the model.

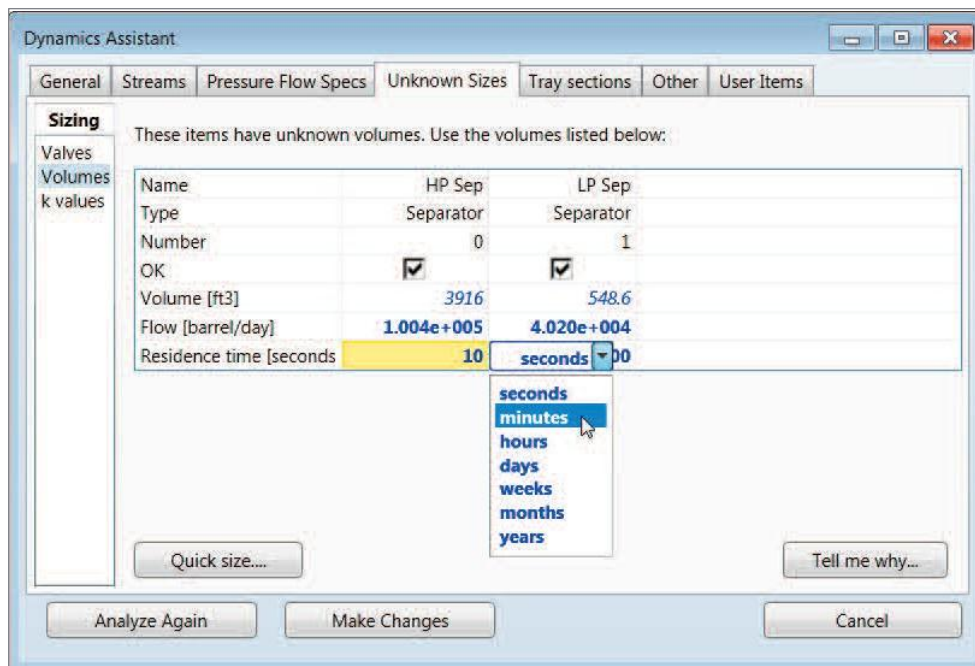


Figure 12: Set the residence time to determine vessel volume

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

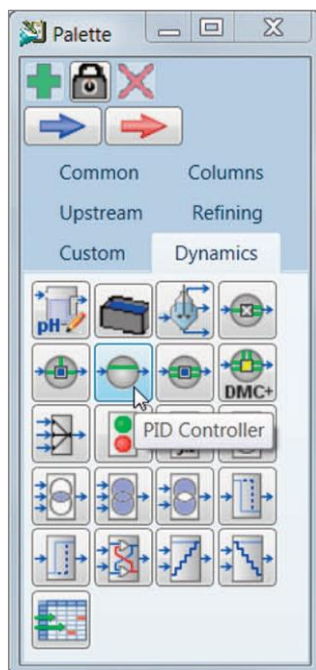


Figure 13: The Dynamic tab of the Model Palette



## 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 dramatically impact an entire system. It is important to establish a control scheme before running your simulation dynamically.

### Controllers on the Model Palette

Aspen HYSYS provides a variety of controllers and logical operations that you can use for your control scheme. These are all available on the Dynamic tab of the Model Palette (accessible from the View tab on the ribbon). Hover over any icon to see what controller it represents and find the one that you need (Figure 13).

### 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

Instantiate a PID controller and double click on it to view its properties form. On the Connections tab, select the Process Variable and Output Target Object (Figure 14). The process variable is the actual plant value that the controller is trying to regulate. The output target object (sometimes referred to as the manipulated variable), is the parameter that the controller will change to adjust the process variable to the set point. The controller below (from Case 2) is using the actuator position of the valve VLV-HP Vap to control the pressure in the column HP Sep.

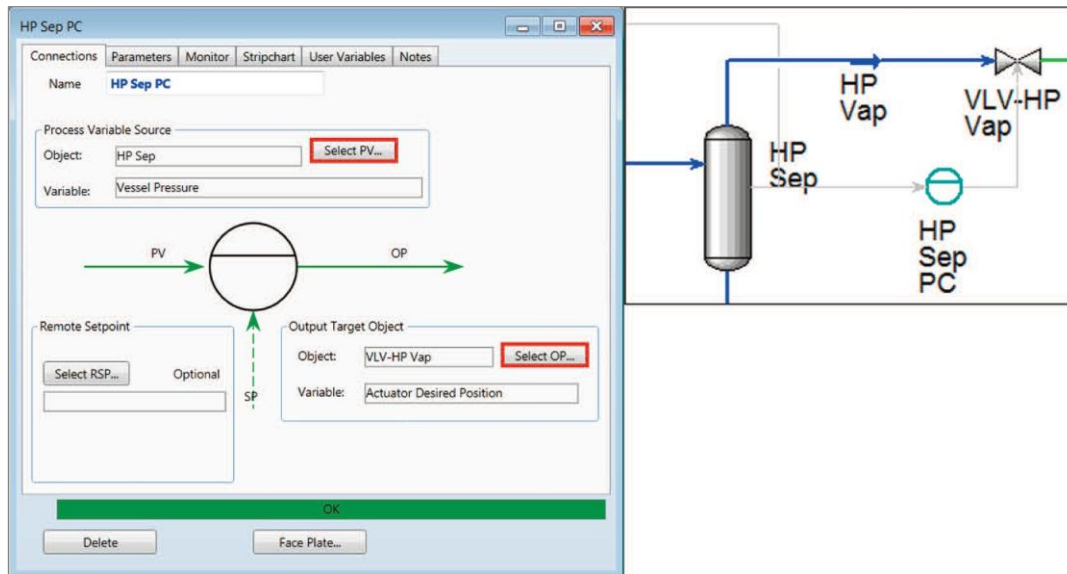


Figure 14: Select a process variable and output target object for a PID controller

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.” To understand the difference between direct and reverse action, consider Figure 15: 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 15 explains the same principle if the tank level rises.

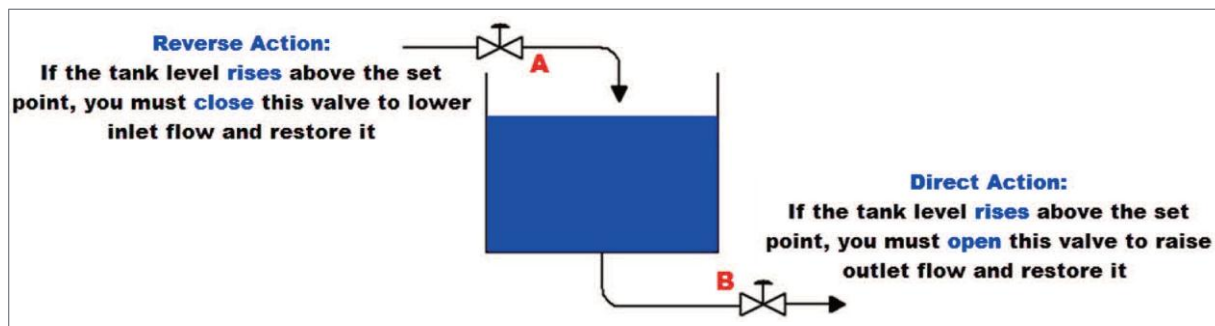


Figure 15: Direct vs. reverse controller action

Next, specify the set point and the PV Minimum and Maximum values. Finally, set the tuning parameters (see the recommendations on Table 2 on the previous page). Figure 16 shows the Parameters tab for the same PID controller and highlights all of the inputs you must specify. Finally, you can activate the PID controller by switching the Mode to “Auto” (in the Operational Parameters section).

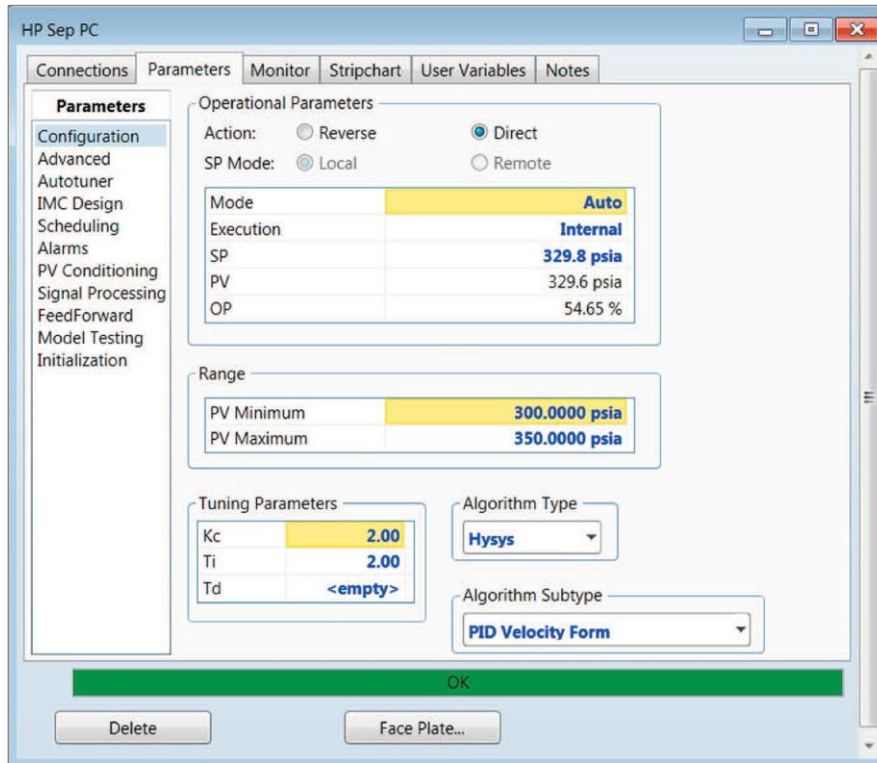


Figure 16: The Parameters tab for a PID controller

## Face Plates

For every controller, you can view a face plate. Either click 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 17).

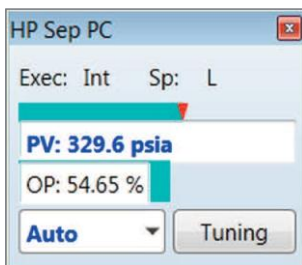


Figure 17: Controller face plate: move the red arrow to adjust the set point and see how the process variable and controller output respond while the simulation is running

A list of all controllers can be quickly found using the Control Manager button in the Tools group on the Dynamic tab (Figure 18).



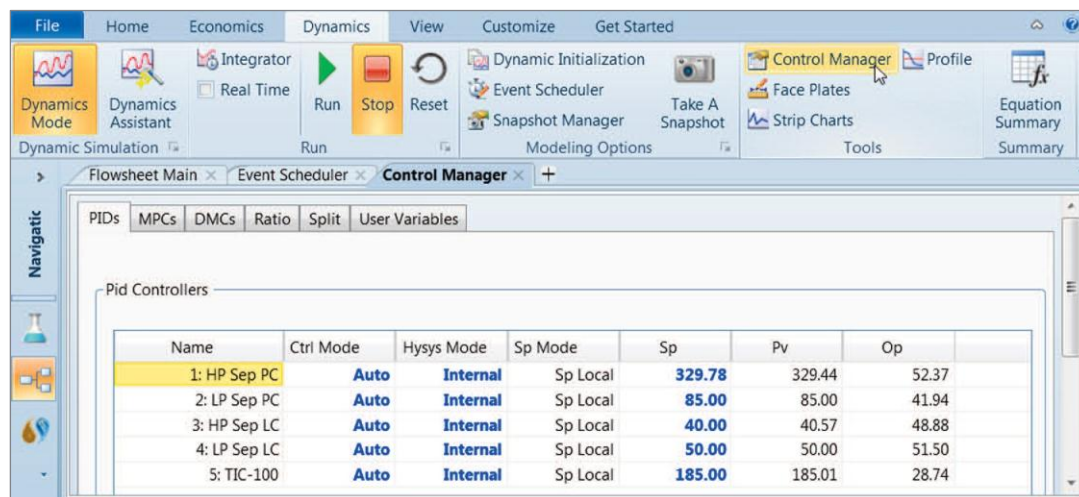


Figure 18: The Control Manager

Likewise, a list of all face plates can be found using the Face Plate button (Figure 19).

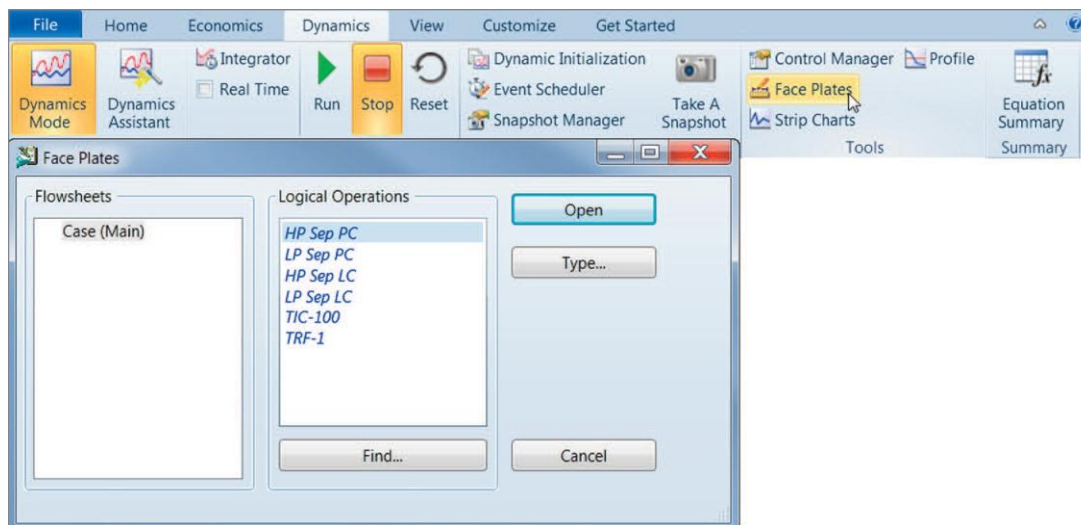


Figure 19: View all face plates

## 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 Dynamic Mode. Use the Dynamic Assistant to make sure that you are not missing any important specifications and then click the Dynamic Mode button (Figure 20).

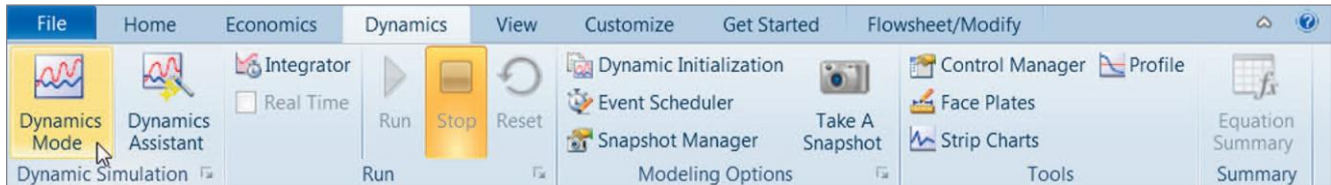


Figure 20: The Dynamic Mode button

## Strip charts

Strip charts are a way to observe the dynamic fluctuations of you 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 Dynamic tab. Open the Strip chart page to see several default variable sets. Select one from the Variable Set dropdown and click the Create Stripchart... button (Figure 21). For controllers, there is a Strip chart tab.

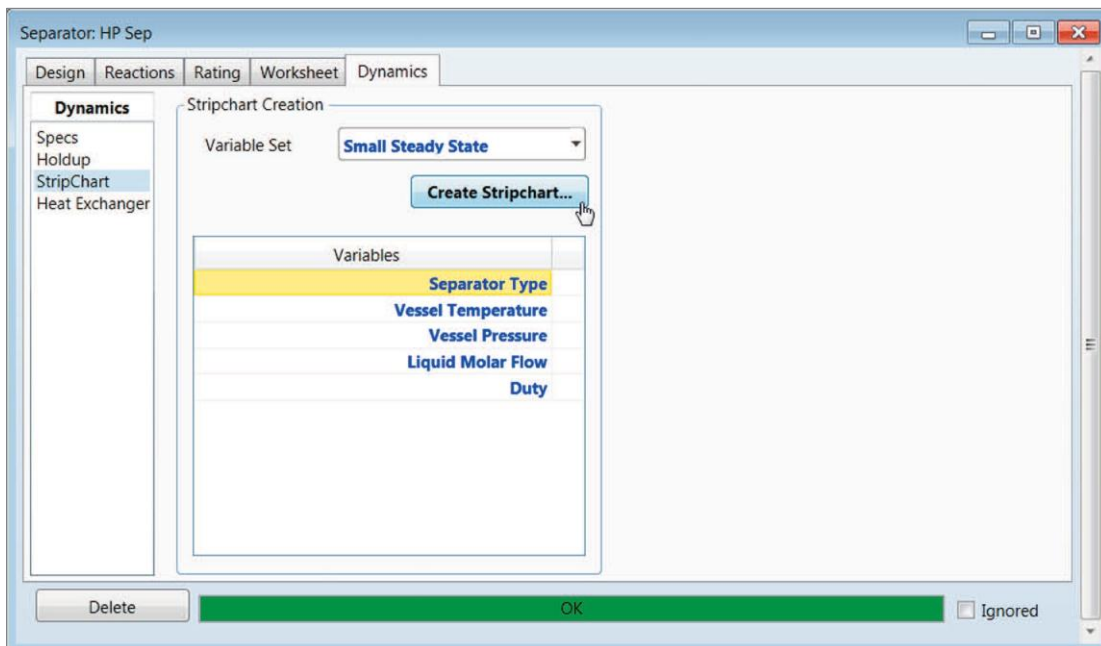


Figure 21: Generate a strip chart for a unit operation using a default variable set

You can also generate a custom strip chart using the Strip Charts button in the Tools group on the Dynamic tab (Figure 22). This will open the StripChart window.

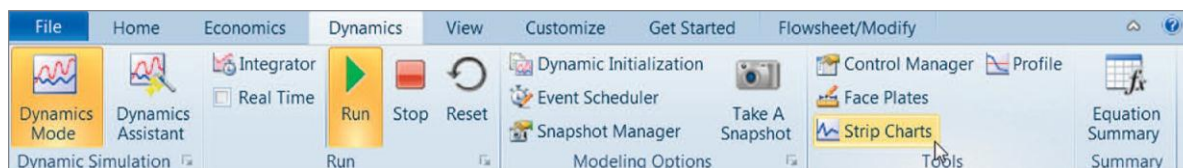


Figure 22: The Strip Charts button

Click the Add button to create a new DataLogger which logs the information necessary to generate the strip chart (Figure 23). The images below show how the existing strip chart HP Sep PC-DL1 was created in Case 2.

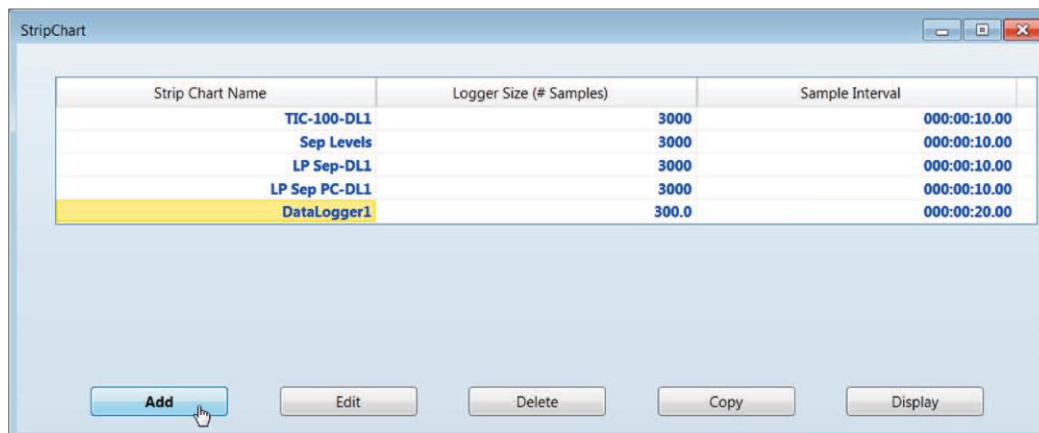


Figure 23: Add a new DataLogger

Change the name to something more descriptive in the Strip Chart Name column. Also, change the Logger Size and Sample Interval. Once any cell in the row is selected, click the Edit button (Figure 24).

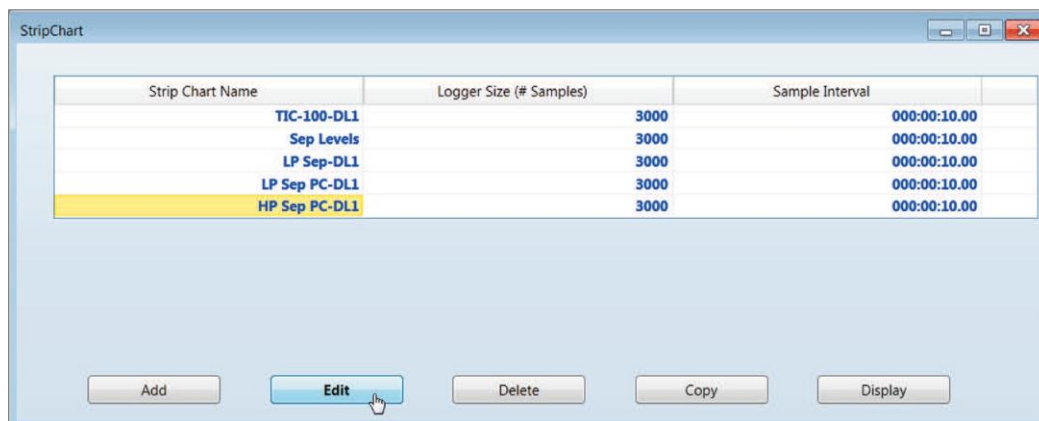


Figure 24: Edit a DataLogger

In the window that appears, click on the Add button to add the variables that you want to track (Figure 25). You can deactivate any of these later by clicking the checkbox in the Active column on the right. You can also edit or delete them using the buttons at the bottom.

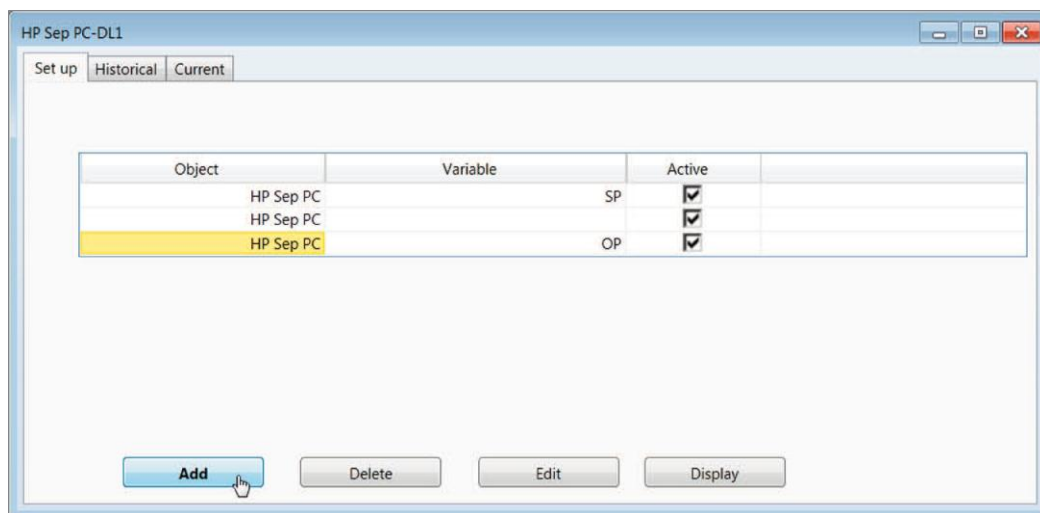


Figure 25: Add new variables to track on a strip chart

To generate the chart, click the Display button. If you are in Run mode (the Run button on Dynamic tab is selected), the chart will automatically start populating. It is possible to scroll backwards, forwards, up, and down on a strip chart by clicking on the graph space and dragging it.

### Formatting a Strip Chart

To format a strip chart, right-click on the plot area and select “Graph Control...” (Figure 26).

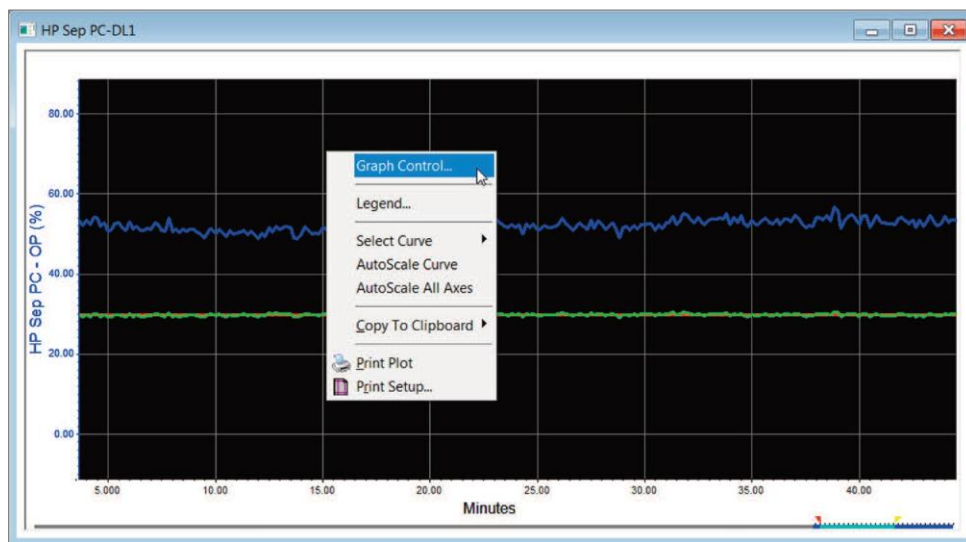


Figure 26: Open graph control for a strip chart

This will open the Strip Chart Configuration window (Figure 27). From here, you can adjust how each variable is presented, the axes, and other display parameters for this particular chart.

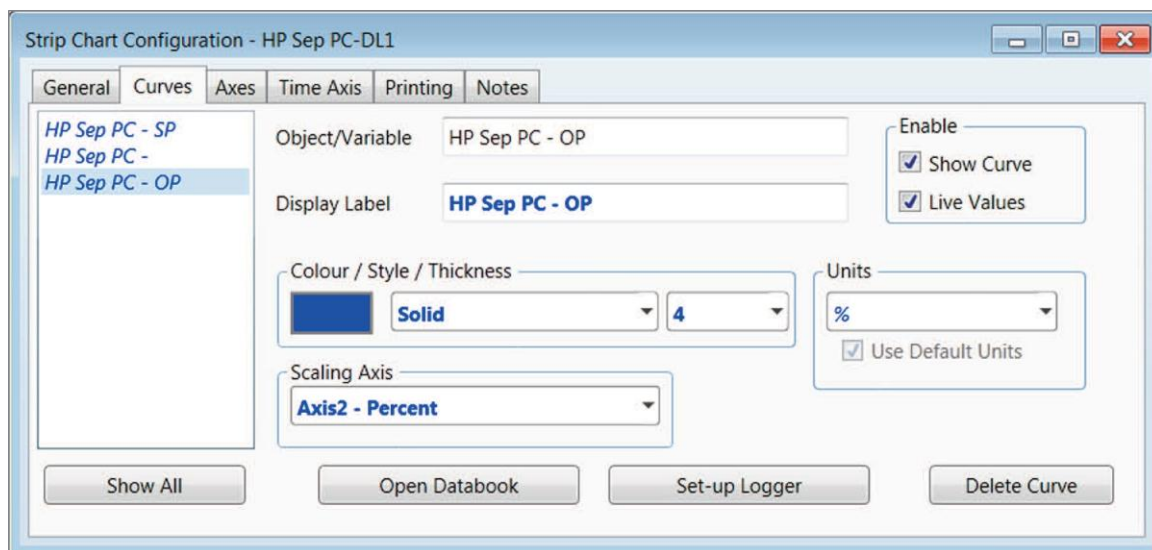


Figure 27: Strip Chart Configuration pane

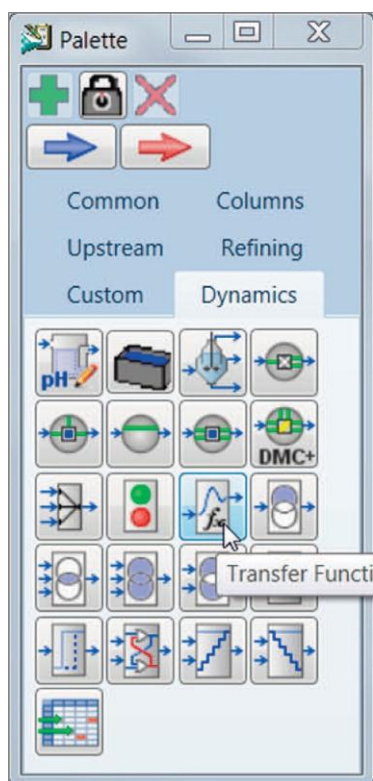


Figure 28: The Transfer Function icon

## 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 add a transfer function, select the Transfer Function block on the Dynamic tab of the Model Palette (Figure 28).

Once you have instantiated a transfer function, open its property form. From the Connections tab, you can select the target stream, unit operation, or controller that you want the transfer function to affect. You can also optionally select a process variable to guide the transfer function's behavior. Figure 29 shows a transfer function that acts on the stream NGL feed in Case 2.



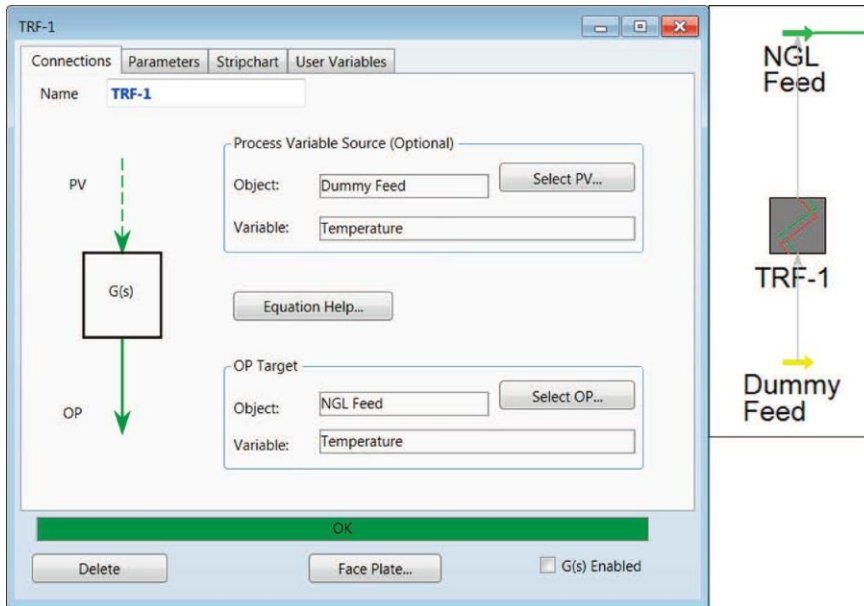


Figure 29: Select the target for a transfer function

On the Parameters tab, you can fill out the remaining inputs for your transfer function. You must fill in the Ranges section before you can fill in the Operational Parameters (Figure 30).

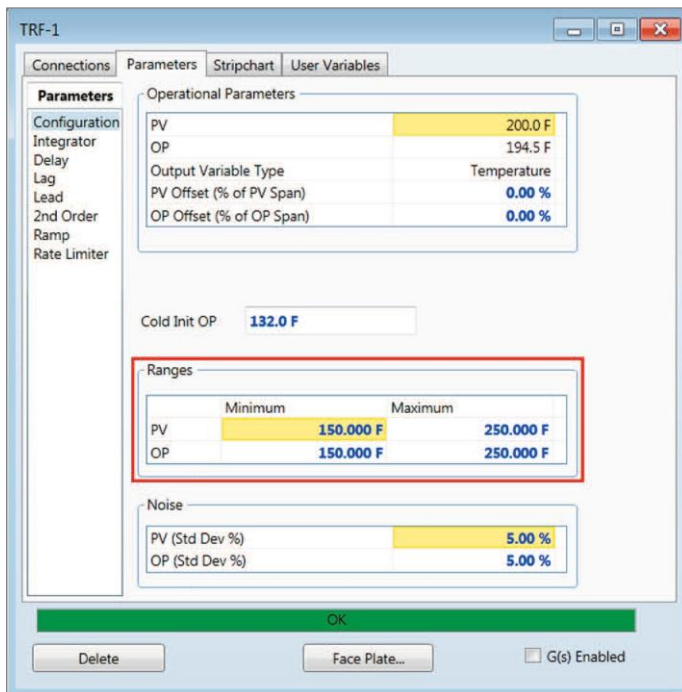


Figure 30: Specify the parameters for a transfer function – you must fill in the ranges first

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 Aspen HYSYS to 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.

The images below show how the existing Schedule 1 was built in Case 2.

### Add a Sequence

Click the Event Scheduler button in the Modeling Options group on the Dynamic tab (Figure 31).

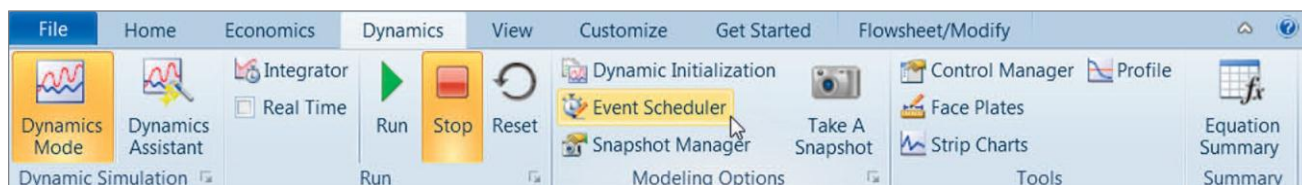


Figure 31: The Event Scheduler button

Next, click the Add button in the Schedule Options section (Figure 32).

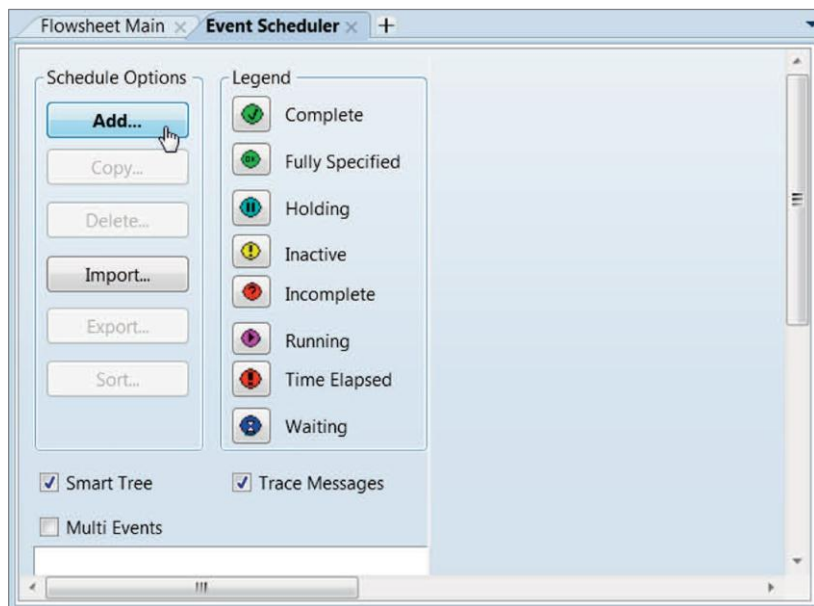


Figure 32: Add a schedule

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 (e.g., FeedTChange in the Figure below) and determine if you want to act in one shot or continuously. Next, select the sequence and click the View... button (Figure 33).



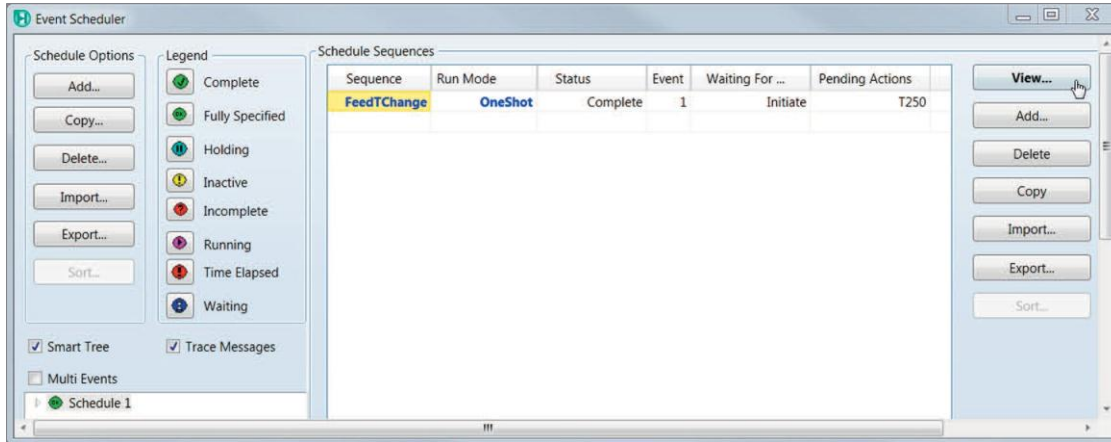


Figure 33: View a sequence

### Add an Event

In the Sequence window 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, select the event and click View... (Figure 34).

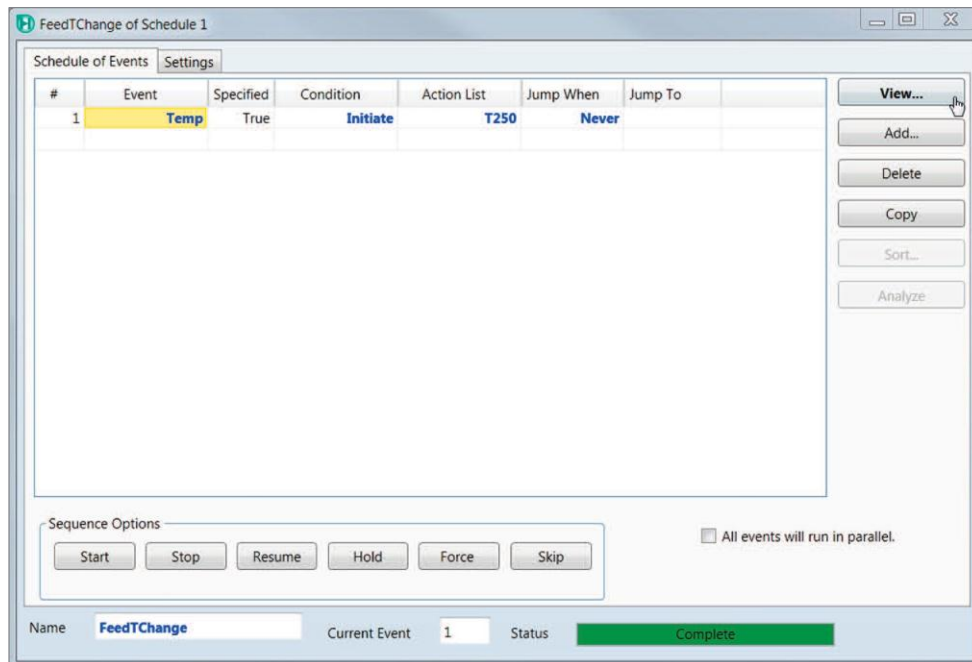


Figure 34: Edit a specific event

On the window that follows, you can specify how you want the event to be triggered on the Condition tab.

### 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.

Case 2 shows a unique application of logical expressions. In this case the expression is "1=1," which will immediately evaluate as true. This will instantly activate the event once the sequence has started (Figure 35).

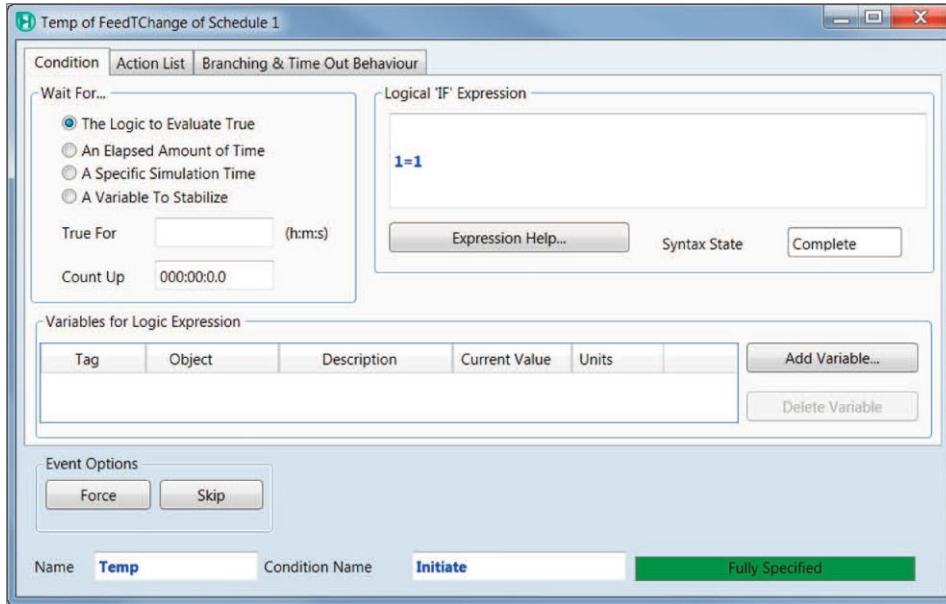


Figure 35 shows the 'Condition' tab of the 'Temp of FeedTChange of Schedule 1' window. The 'Wait For...' section has four radio buttons: 'The Logic to Evaluate True' (selected), 'An Elapsed Amount of Time', 'A Specific Simulation Time', and 'A Variable To Stabilize'. Below these are fields for 'True For' (h:m:s) and 'Count Up' (000:00:0.0). The 'Logical 'IF' Expression' field contains '1=1'. Below this is a table for 'Variables for Logic Expression' with columns: Tag, Object, Description, Current Value, Units. There are 'Add Variable...' and 'Delete Variable' buttons. At the bottom, there are 'Event Options' (Force, Skip) and a status bar showing 'Name: Temp', 'Condition Name: Initiate', and 'Fully Specified'.

Figure 35: Trigger the event immediately using logical expressions

## 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.

In Case 2, there is one action taken, which changes the feed temperature to 205°F (Figure 36).

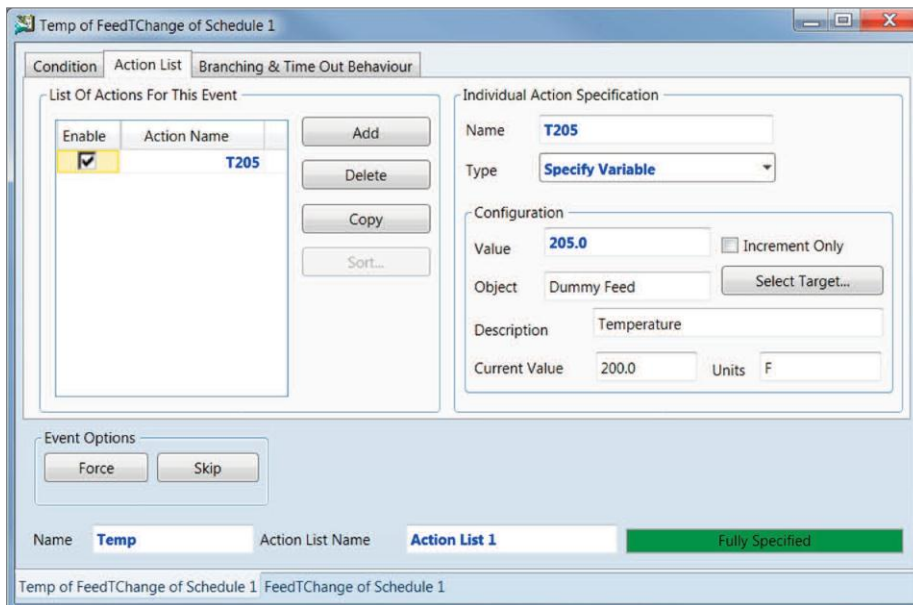


Figure 36 shows the 'Action List' tab of the 'Temp of FeedTChange of Schedule 1' window. The 'List Of Actions For This Event' section has a table with columns: Enable, Action Name. There is one row with 'T205' and a checked 'Enable' box. There are 'Add', 'Delete', 'Copy', and 'Sort...' buttons. The 'Individual Action Specification' section has fields for 'Name' (T205), 'Type' (Specify Variable), 'Value' (205.0), 'Object' (Dummy Feed), 'Description' (Temperature), and 'Current Value' (200.0). There is an 'Increment Only' checkbox and a 'Select Target...' button. At the bottom, there are 'Event Options' (Force, Skip) and a status bar showing 'Name: Temp', 'Action List Name: Action List 1', and 'Fully Specified'.

Figure 36: Add actions and configure specifications

## 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 37). 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.

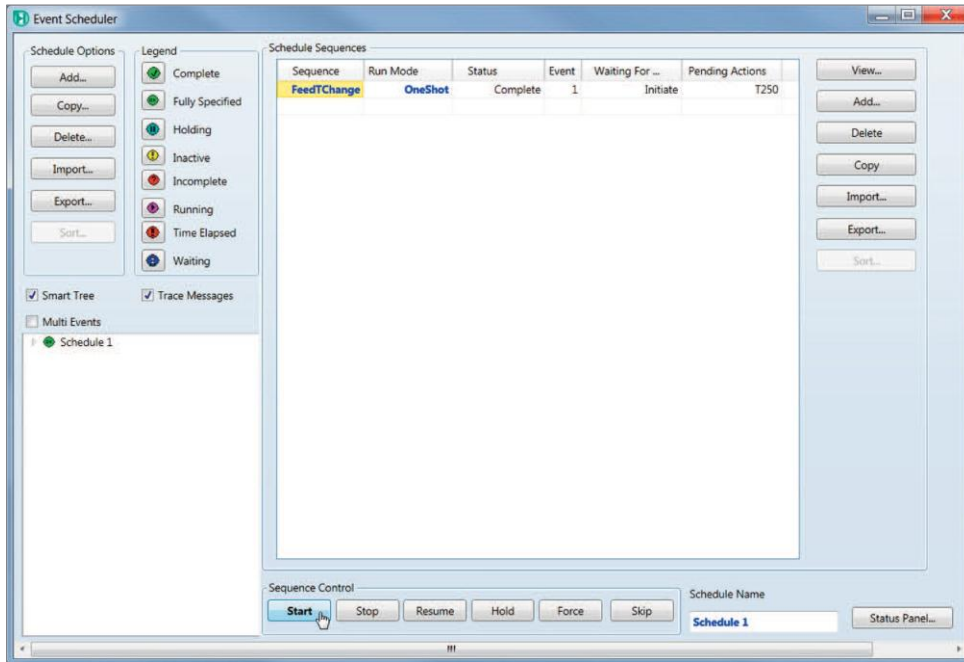


Figure 37: Start a given sequence

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