

Jump Start: Aspen HYSYS® 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

Introduction
Start with a Steady State Model2
The Dynamics Assistant
Specify Pressure and Flow Parameters
Pressure Node and Resistance Equation Operations
Boundary Stream Pressure Specifications5
Dynamics Color Scheme
Size the Equipment
Design a Control Strategy
The Controller Library
The PID Controller
Setting a PID Controller
Face Plates
Running Your Model
Strip Charts
Generating a Strip Chart
Formatting a Strip Chart
Transfer Functions
Event Scheduler
Add a Sequence
Add an Event
Using Logical Expressions
Specify Event Actions
Activating Sequences
Additional Resources

Introduction

Dynamic modeling of process simulations is vital to plant design and operation. Although there are many scenarios in which steadystate 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) 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).

Contraction and the						Window Help								
D	2	2 🍅 🛛	Q. « 🗉) #1	ц,		🔤 D.	≪ a [□]	小圈	2	Ш	ò	()	4

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).

		Make changes
	Disable stream flow specifications	 Image: A second s
	Enable stream pressure specifications	1
	Append new valves and streams	V
	Enable pressure flow equations not pressure drop	1
	Miscellaneous specification changes	1
	Volumes not known	1
	Pressure flow k values not known	~
A	oout Preferences □ □ Sa	ve steady state case

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).

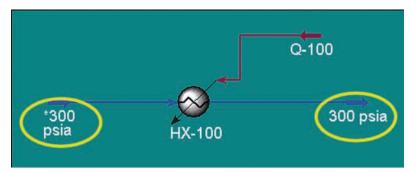


Figure 3: You must specify a pressure drop across resistance operations to run a dynamic simulation

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.

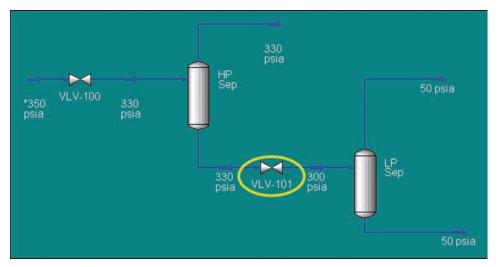


Figure 4: You must have a resistance operation between every pressure node, in this case a valve between two columns

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).

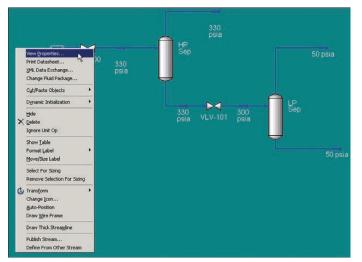


Figure 5: Open the Properties form for a stream

Material Strea	m: NGL Feed	Material Stream: NGL Feed	
Dynamics Specs Stripchart	Dynamic Specifications Pressure Specification 350 psia Flow Specification Molar Mass Molar Mass Ideal LiqVol Std. LiqVol Ideal Liquid Volume Flow Active 3.019e+004 barrel/day Image: Content of the second	Dynamics Dynamic Specifications Stripchart Pressure Specification Flow Specification Image: Comparison of the system of th	Noi
Worksheet	Attachments Dynamics	Worksheet Attachments Dynamics	
Delete	Define from Other Stream 💠 🜩	Delete Define from Other Stream	* *

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

© 2015 Aspen Technology, Inc. AspenTech", aspenONE*, the Aspen leaf logo, the aspenONE logo, and OPTIMIZE are trademarks of Aspen Technology, Inc. All rights reserved.11-7689-0715

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.

C PFD - Case (Main)			
	HW	ρA	. 🌮 🎕	Dynamic P/F Specs Default Colour Scheme
				 Default Colour Scheme HYSIM Colour Scheme Temperature Dynamic P/F Specs

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.

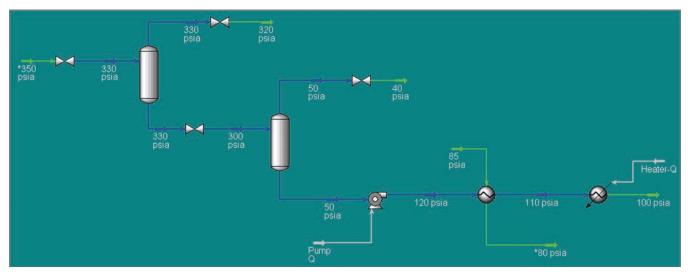


Figure 8: PFD with appropriate pressure and flow specifications for dynamic simulation

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).

Rating	Geometry		~	0.446.044
Sizing		Irientation:	C Horizontal	Quick Size
-	C Sphere	Volume [ft3]	3916	Weir
Nozzles	C Ellipsoidal Head	Diameter [ft]	14.92	
Heat Loss	C Hemispherical Head	Height [ft]	22.39	Enable Weir
Level Taps	. Temsprenca neau	Head height [ft]	<empty></empty>	
Options C.Over Setup C.Over Results	This separator has a	2 <u>0</u> 000		
C.Over Setup		- ⁷ 004		

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).

Sizing	These items have un	known volumes. Us	e the volumes list	ed below
Valves	Name	HP Sep	LP Sep	
Volumes	Туре	Separator	Separator	
k values	Number	0	1	
K values	OK	V	V	
	Volume [ft3]	3916	546.5	
	Flow [barrel/day]	1.004e+005	2.803e+004	
	Residence time	10	seconds 💌	
			seconds	1
	Quick size		minutes hours days weeks months years	e why
General Stream	ns Pressure Flow Spece	Unknown Siz	zes Tray sectio	ns er

Figure 10: Set the residence time to determine vessel volume

Note: For simple heat exchangers, there is a default vessel size of 0.1m³. 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

aspentech

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 at on the lower right. Figure 11 shows the location of these buttons and the resulting palettes.

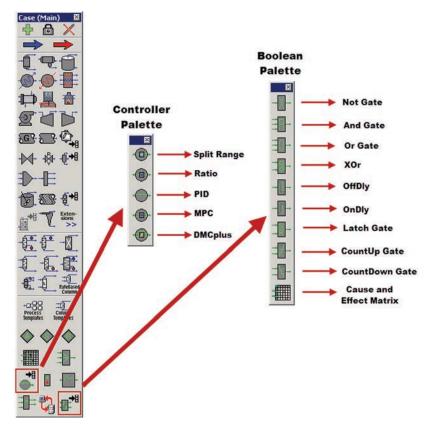


Figure 11: The Control Ops and Boolean Ops Palettes and their constituents

The PID Controller

aspentech

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, τ_i , and the derivative gain, τ_d . Table 2 shows our suggestions for initial PID tuning parameters. Note that τ_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	Kc	τ _i (minutes)	τ_{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.

Name IC-100					
Process Variable Object: Variable:	Source		Select PV.]	
Remote Setpoint	Dptional		etput Target 0 Object: Variable:	OP bject	Select OP
Connections	Parameters	<u>Monitor</u>	Stripchart K	User Variables	Notes

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 (aspentech

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.

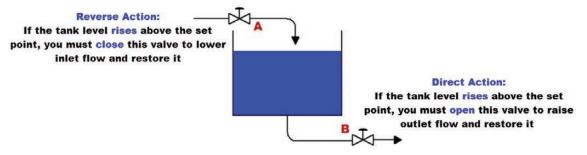


Figure 13: Direct vs. reverse controller action

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".

Parameters	Operational Parameters	
Configuration	Action: Preverse C Direct SP Mode: Local C Remote	
Advanced	Mode Off	
Autotuner	Execution Internal	
IMC Design	SP <empty></empty>	
-	PV <empty></empty>	
Scheduling	OP 50.00 %	
Alarms	Range	
PV Conditioning	PV Minimum (empty)	
Signal Processing	PV Maximum (empty)	
FeedForward	LT	
Model Testing	Tuning Parameters Algorithm Type	
	Kc 1.00 Hysys 🔻	
Initialization	Ti <empty></empty>	
	Td Algorithm Subtype	
	PID Velocity Form	
		_
Connections P	arameters Monitor Stripchart User Variables Notes	
	Unknown Ranges for PV	

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).

PID Controll	er: HP
Exec:Int Sp	b: L
PV: 329.62 p	
0P: 54.65	
Auto 💌	Tuning

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 [x] in the toolbar (Figure 16).

a strategies			Flowsheet			100000000									
D	2	🖬 🍃	0, 08	. #	हिः 🖸	1 1 I	₿. •	€ a□	f 🕮	27 🛍	=	×	X	1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	4

Figure 16: The Dynamics 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 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.

Separator: HP Sep		
Dynamics	Stripchart Creation	
Specs	Variable Set Small Dynamic State ▼	
Holdup	Create Stripchart	
StripChart	Variables	
Heat Exchanger	Separator Type	
	Vessel Temperature	
	Vessel Pressure	
	Liquid Molar Flow	
	Duty	
	Tank Volume	
	Liquid Volume Percent	
Design Reaction	ns Rating Worksheet Dynamics	
Delete	OK	🔽 Ignored

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).

Available Strip Charts HP Sep PC-DL1 LP Sep PC-DL1 LP Sep-DL1	View Strip Chart	Individual Strip Chart Data Selection	mple Int. 000:00:20.00	
Sep Levels	Historical	Object	Variable	Active
TIC-100-DL1		HP Sep PC	SP	R
	Current	HP Sep PC	PV	A
		HP Sep PC	OP	V
	Add	TIC-100	SP	T
		TIC-100	PV	Г
	Delete	TIC-100	OP	Ē
		LP Sep	Liquid Percent Level	Ē
	Setup	HP Sep	Liquid Percent Level	Ē
	Setup All	LP Sep	Separator Type	Ē
	Detub Mit	LP Sep	Molar Flow	Ē
		LP Sep	Duty	Ē
		LP Sep	Tank Volume	Ê
		LP Sep	Liquid Volume Percent	Ē
		LP Sep PC	SP	Ē
		LP Sep PC	PV	Г
		LP Sep PC	OP	Ē

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.

Lo	gger Set-Up - HP Sep
F	Logger Size (# Samples)
	3000
5	Sample Interval
	000:00:10.00

aspentech

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

💐 HP Sep PC-DL1 _ 🗆 🗙 100.00 Graph Control. 80.00 Legend. Select Curve (%) dO -AutoScale Curve 60.0 AutoScale All Axes 52.79 (%) HP Sep PC ⊆opy To Clipboard Print Plot 40.0 329.8 (bsis) 20.00 0.00 2.422e 2.424e+004 2.423e+004 2.425e+004 Minutes

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.

HP Sep PC - SP HP Sep PC - HP Sep PC - OP	Object/Variable HP Sep PC - OP	Enable Show <u>C</u> urve
	Display Label HP Sep PC - OP Colour / Style / Thickness Solid Scaling Axis Axis2 - Percent	
Show All	Open Databook Set-up Logger	Delete Curve

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 for 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).

Transfer Func	tion Block: TRF-2
Name TRF-2	
PV	Process Variable Source (Optional) Object:
G(s)	Eguation Help
OP ¥	Object: Select OP Variable:
Connections	Parameters Stripchart User Variables
	Requires an Input Variable
Delete	Face Plate 🔽 🖸 🤤 (s) Enabled

Figure 23: Select the target for a transfer function

(aspentech

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).

Parameters	Operational Parameters	
Configuration	PV	cempty:
2000.00 <u>7</u> 0.00200.000	OP	<empty:< td=""></empty:<>
ntegrator	Output Variable Type	Unitless
elay	PV Offset (% of PV Span)	<empty:< td=""></empty:<>
a	OP Offset (% of OP Span)	<empty:< td=""></empty:<>
amp	Reset out of range PV value using s Cold Init OP <empty></empty>	specified range j
		specified range
ind Order Iamp Iate Limiter	Cold Init OP <empty></empty>	specified range j
amp	Cold Init OP <empty></empty>	Maximum
amp	Cold Init OP <emply> Banges</emply>	
imp	Cold Init OP <empty> Banges</empty>	Maximum
amp	Cold Init DP <empty> Eanges Vinimum PV <empty></empty></empty>	Maximum <empty:< td=""></empty:<>
amp	Cold Init OP <empty> Banges PV <empty> OP <empty></empty></empty></empty>	Maximum <empty:< td=""></empty:<>

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).

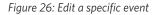
Add 🛛 🐼 Complete	Sequence	Run Mode	Status	Event	Waiting For	Pending Actions	⊻iew
	Sequence A	OneShot	Incomplete				
opy Fully Specified Holding							<u>A</u> dd
siete							<u>D</u> elete
port							Сору
port Running							Import
ort 🐠 Time Elapsed							Export
Waiting							Sort

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.

‡	Event	Specified	Condition	Action List	Jump When	Jump To	⊻iew
1	Event 1		Condition 1	Action 1	Never		
		45					<u>A</u> dd
							<u>D</u> elete
-							Сору
							Sort
-							Analyze
eque	nce Options					All events will ru	n in parallel.
St	art Stop	Resume	Hold	Force	ikip		
Sche	dule of Even	ts Settings					



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.

C An Elaps C A Specifi	c to Evaluate True ed Amount of Tim c Simulation Time e To Stabilize	e	/ar1==90	000			
True For		(h:m:s)	E <u>x</u>	pression Help	Sj	untax State	Complete
Journop	ogic Expression						
Tag	Object	Descripti	on	Current Value	Units		dd Variable
Var1	NGL Feed	Actual Volur	ne Flow	8.905e+004	barrel/day		
					4		elete Variable
	Action List	Branching & Tir	me Out B	ehaviour			
Condition)			
condition							

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).

		Action Name Close VLV-100	Add Delete Copy Sort	Name Type Configur Value Object Descript Current	Close VLV-100 Specify Variable Start Sequence Play Sound Trace Dump Hold Sequence Play Script Save Snapshot Send DDE Command Stop Integrator Ramp Controller Set Controller Mode	ement Only ct Target
Condition Action List Branching & Time Out Behaviour	Condition	Action List	Iranching & Time	Out Behaviour		

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

aspentech

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.

V Complete	Sequence	Run Mode	Status	Event	Waiting For	Pending Actions	⊻iew
🛛 🐵 Fully Specified	Reduce Vol Ir	OneShot	Inactive	1	High Feed Vol	Close VLV-100	<u>A</u> dd
e (1) Holding							Delete
t							Сору
t Incomplete t Running							Import
Time Elapsed							Export
🕘 🕲 Waiting							Sort
hedule 1 • Reduce Vol In							
	-Sequence Control						

Figure 29: Start a given sequence

Additional Resources

For further information on Aspen HYSYS please consult:

Public Website:

http://www.aspentech.com/products/aspen-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

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.



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

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

© 2015 Aspen Technology, Inc. AspenTech^{*}, aspenONE^{*}, the Aspen leaf logo, the aspenONE logo, and OPTIMIZE are trademarks of Aspen Technology, Inc. All rights reserved. 11-7689-0715